READ 2024
RESEARCH & EDUCATION IN AIRCRAFT DESIGN
WARSAW, POLAND | 6-8 NOVEMBER 2024

A CFD Investigation of an UAV Fixed-pitch Rotor in Flight Regime Transition

Edmund Havran[1]

[1]Faculty of Mechanical Engineering, Institute of Aerospace Engineering, Brno University of Technology, Brno, Czech Republic

Abstract

The growing popularity and relevance of multicopter type UAVs in recent years have brought aerial vehicles closer to human lives than ever before. This is causing a pressure to lower the noise of the vehicles to more pleasant levels for human hearing. Designing a rotor with even a small decrease of aerodynamic noise could create a large competitive advantage for the UAV manufacturers.

Similarly to helicopters, majority of UAVs rely on their rotors to provide lift during the flight. But unlike helicopters which use variable-pitch blades on the rotors for flight control, multicopter type UAVs typically use differential thrust on their rotors to provide flight control. Rotors then have only fixed-pitch rotor blades. Over these blades, the flow changes are chiefly caused by changes in rotational speed, and changes in flight regimes. A typical UAV flight envelope involves many different flow conditions on its rotor blades with flight manoeuvres such as transition from hover to forward flight and back, manoeuvring at higher speeds in small spaces etc. The goal of this paper is to investigate these transitional phases to open up new possibilities of rotor blade improvement, as a large part of the rotor aerodynamic noise is associated with local stalls on the blades of the rotor that can be caused by dynamic flow condition changes.

This paper introduces a two-bladed fixed-pitch rotor in accordance with Caradonna and Tung [12], the so-called C-T rotor, in a large sphere domain simulated in the Ansys Fluent CFD software. A similar Reynolds number flow to larger multicopter UAVs is used. First the simulation is calibrated using a mesh dependency study in the hover flight condition in accordance with experimental measurements. Next a transient simulation is calculated with changing flow velocity and direction on the domain boundary, simulating the transition of the UAV flight regimes between hover and forward flight. An investigation on the flow behaviour near the blades during the transition and on the thrust, moment and acoustic noise of the rotor is done. This helps quantify the influence of flight regime transition on the possible performance of a multicopter UAV.

Keywords: rotor, aerodynamics, acoustics, transient rotor noise

1. Introduction

The world is currently experiencing a large increase in relevance of small multicopter unmanned aerial vehicle (UAV) of increasing variety. This has been also augmented by development in urban air mobility (UAM) vehicles. The ever-larger application for tasks closer to human lives, such as drone delivery [1] will necessitate the introduction of legislative to control and lower the noise these aircraft would generate. As even a small decrease in dB values could be influential in attaining a competitive advantage for the drone manufacturer, a lot of research has been done in this field in the last couple of years. The rotor designer of the future will necessarily need to design a rotor that will provide as low of an acoustic noise as possible, while also improving the rotor performance. The goal is then to provide the knowledge and methods for this while also keeping costs down.

Computational fluid dynamics (CFD) might offer an easy solution but they depend heavily on the chosen method to assure a sufficient accuracy.

1.1 Background

Most of the work on rotary wing acoustics has been done on helicopters and propellers. A short summary of the historical trends can be found in [2] and [3]. This research is still relevant in the smaller scale UAV and UAMs, but one must consider the differences of a helicopter flight as it usually makes only relatively gentle transitions between flight regimes, limited both by the higher weight and thus momentum and by the human factor, as the pilots or passengers would not be comfortable with many high-G manoeuvres. The small size of UAVs coupled with their majority fixed pitch multirotor configuration provides flight regime changes in much more sudden way. The extreme end of this is represented by the FPV drones, that are also the smallest and thus the cheapest and most prevalent type of currently available UAVs. This type of UAV changes its flight trajectory with violent high G manoeuvres that are created by changing the motor speeds on their rotors. Pitch and roll motions are ensured by lowering the RPM of rotors on one side while increasing it on the other side. As this happens, the rotors undergo either an acceleration or deceleration in their rotational speed while also changing the inflow angle. This creates potential for flow disturbances, the more sudden a manoeuvre the more pronounced. As transition regimes make up the smallest part of UAV flight time, they are not as important from a designer's point of view than hover and forward flight. But these transitions mainly happen low above the ground, for example while landing or taking off and thus can be important for lowering UAV noise.

1.2 Noise in UAVs

On the topic of noise generation in UAVs, there has been both experimental and computational simulation research done. At first lessons from helicopters were used but especially in recent years research focused only on UAVs has appeared. One of the first major works in angular inflow conditions is the work done by NASA in [4]. This paper introduced a new theory and computational program to predict the noise of propellers. Experimental verification showed quite good accuracy. In more recent years, research with special focus on manoeuvring flight noise has appeared. The goal was to develop strategies for helicopter pilots to know the noise generated by their flight and thus lower it by careful flight management such as in [5] and [8], but also more general prediction methods for engineers such as in [6] or [7]. Next step was to implement more unsteady variables and CFD methods [2] to provide an accurate prediction of both tonal and broadband noises and investigate their sources including helicopter hull interference and blade-vortex interaction. In more recent years UAVs themselves were the subject. Many works investigated edgewise flow noise both in CFD and in experimental setups. [3] provided investigations with a fixed flow direction during the measurement but changing variables like rotor speeds, thrust. This was also investigated in [9] or [10]. In conjunction with test data from hover and forward flight [11], the general conclusion can be made, that changing the inflow angle of air to a rotor will change its noise characteristics. This will both happen because of the flow properties itself but also because of lower rotor thrust generated in such conditions and thus a need for higher rotor speeds (which also increases motor noise).

2. Methods

2.1 Preparation of geometry

As the geometry, a two bladed fixed-pitch rotor using NACA 0012 airfoils is chosen as described in [12]. This so called Caradonna-Tung rotor is a benchmark case in helicopter rotary wings. The rotor is modelled just as two blades without a hub to lower the cell count. This is expected to bring a degree of inaccuracy, but for future simulations with other blades can also mean easier opportunity to simulate just the blades without the hub for different motor mountings. The pitch angle is chosen as 8°. The dimensions of the domain are introduced in table 1 and a visualization is on figure 1. The large size ensures that the flow field will be resolved before it reaches the outer boundary.

Table 1 – Dimensions of the domain.

Rotor diameter - D	2286	mm
Blade chord - c	377	mm
Hub diameter	320	mm
MRF diameter	1.5·D	-
MRF height	2·c	-
Body of influence diameter	2.2·D	-
Domain diameter	17.5·D	-

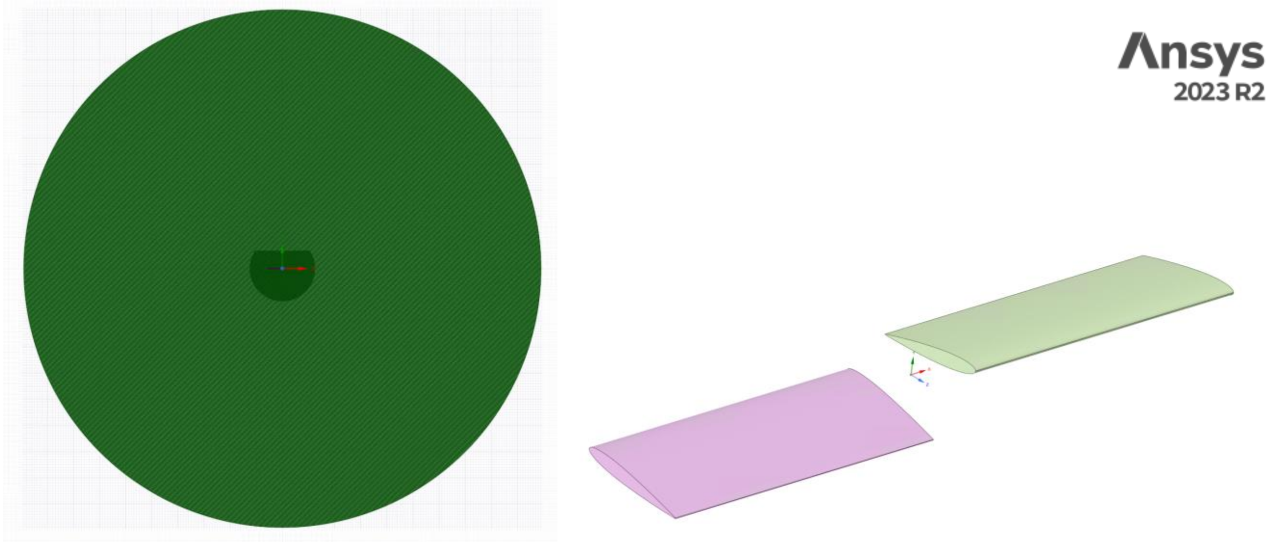

Figure 1 – Domain and rotor.

As the rotor is vastly larger than rotors used on UAV and UAM vehicles, a slower RPM of 650 was chosen. This ensures that the Reynolds number will be in approximately the same range as a larger UAV rotor has.

2.2 Calculation method choice

The software chosen for the simulation is Ansys Fluent. This CFD was chosen because of its availability for the author and implemented usage in the authors institution. It also needs to be said, that Ansys has a dedicated CFD solver for rotating machinery that was not chosen as Fluent is better suited for more complex tasks involving changes in rotor geometry, optimisation, introducing nonrotating body parts such as aircraft hull, and interference between multiple rotors, all of these being use cases that are expected to be encountered in development of new UAVs. Fluent has different methods for rotor simulation but only two are relevant for the given problem: moving reference frame (MRF) and sliding mesh modelling (SM). [13] MRF works by assigning a rotational velocity to the fluid in a designated zone, which essentially simplifies the problem to a quasi-steady solution vastly decreasing the required computational resources. SM on the other hand directly moves the mesh and as such will always be more accurate. [13] The methods themselves also require different approaches to compute, as SM involves a mesh interface. To correctly compute with SM, a fine mesh needs to be created on the boundary between the rotating and stationary parts of the domain. Computations can and will diverge if the boundary is not fine enough and this will usually happen only after a certain time has elapsed, as the boundary cells start in an aligned state in the beginning of a transient simulation for a well-structured mesh. As the time increases, these cells move relative to each other and can cause high skewness on the boundary. For applications that involve a high number of simulations such as rotor optimisation this creates a problem of high risk of the automated optimisation to diverge and crash. On the other hand, the MRF method only requires a correctly defined fluid zone in the domain with just an internal type of boundary. As the mesh itself doesn't move, the cells will stay in the same relative position that can be ensured to be good and the mesh itself doesn't need to be as fine. This vastly decreases the number of cells and thus speeds up the computation. The disadvantage is that the MRF approach can't fully accurately simulate the flow inside the rotating zone. The question is if it is possible to mitigate this accuracy issue for the problem of flight transition enough to be relevant for future calculations.

2.3 Mesh dependency study and current state of work

The program Ansys Fluent Meshing was used for mesh generation. As this program is part of the Fluent package, it doesn't require an additional licence to use, and it is also possible to create a single scripting document that controls both meshing and computation in the same programming language. 3 mesh refinement settings were used for both MRF and SM methods. The refinement of the rotor blades was the same for the respective methods, but the SM mesh needed to be further refined near the boundary. As visible on table 2, this increased the cell count by huge amounts. Because of the limited available computational resources, a difference of multiple millions of cells is very notable. In raw computational time this can mean entire days of CPU-hours saved. Because of this the MRF method was chosen as the faster return time of the simulations was advantageous for simulation setup.

Table 2 – Meshes.

Mesh name	Cell count in millions of cells	Min. orthogonal quality before mesh improvement
MRF		
Mesh 1	1,69	0,21
Mesh 2	1,25	0,12
Mesh 3	0,89	0,07
SM		
Mesh 1	7,41	0,21
Mesh 2	5,70	0,16
Mesh 3	4,97	0,08

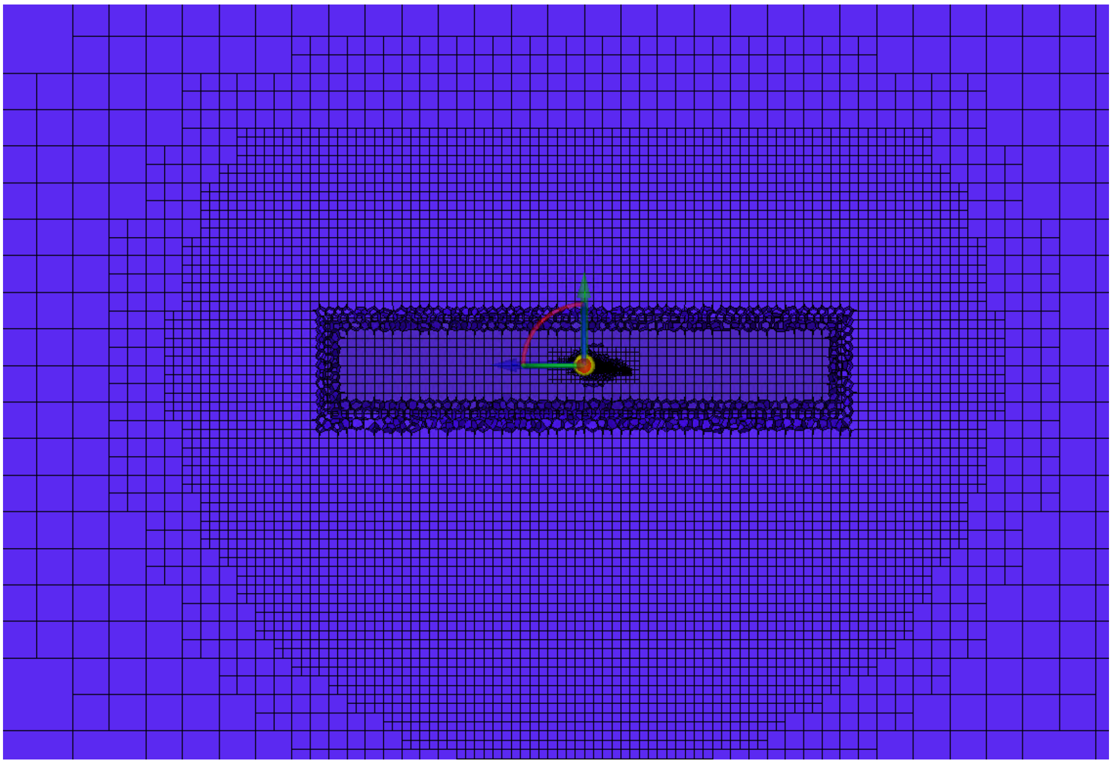

Figure 2 – Mesh detail near the rotating zone.

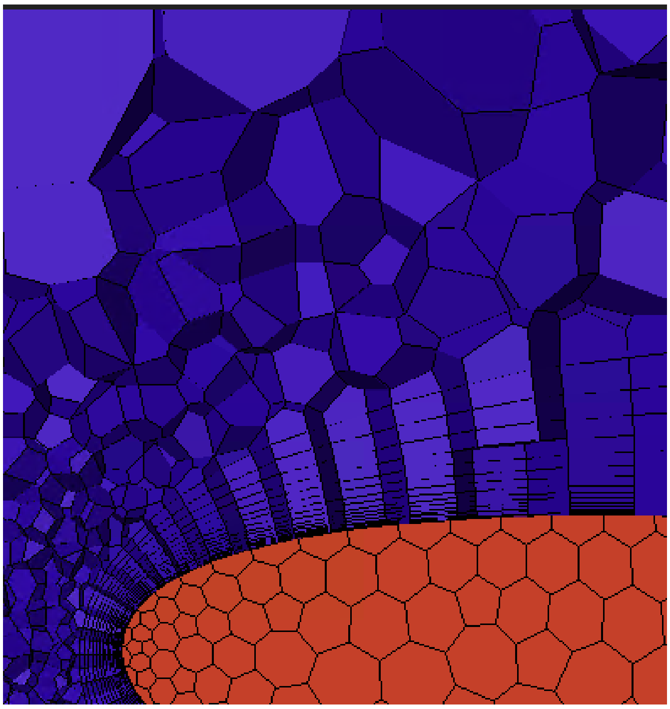

Figure 3 – Mesh detail near the boundary edge.

In Fluent, the domain wall was set up with pressure far field boundary condition. This means that the solver needs to be in density mode and the flow needs to have a non-constant density. The advantage of the pressure far field condition is, that the inlet velocity and direction can change transiently and is not dependent on the location of the inlet and outlet pair, thus the possibility of simulating complex 3D manoeuvres opens up. The simulation was set up in transient mode. Time step size needs to be chosen so as to fulfil the CFL condition, this means a courant number of around 1, but choosing implicit timestep formulation can ensure accuracy even at courant numbers of up to 5. [13] As the reference rotor was measured in hover conditions, free stream velocity is set as 0. The turbulence model was chosen as k-omega SST, as it's the current industry standard and has advantages in comparison with other turbulence models for complex tasks. [13]

This is the current state of work. The computations for MRF have shown a higher degree of inaccuracy than expected, with the pressure coefficient distribution over the blade up to 40% different in comparison to the experimental measurements. This can be seen in figure 4 which shows the results of the mesh 2 for coefficient of pressure at 0,8 radius of the rotor blade.

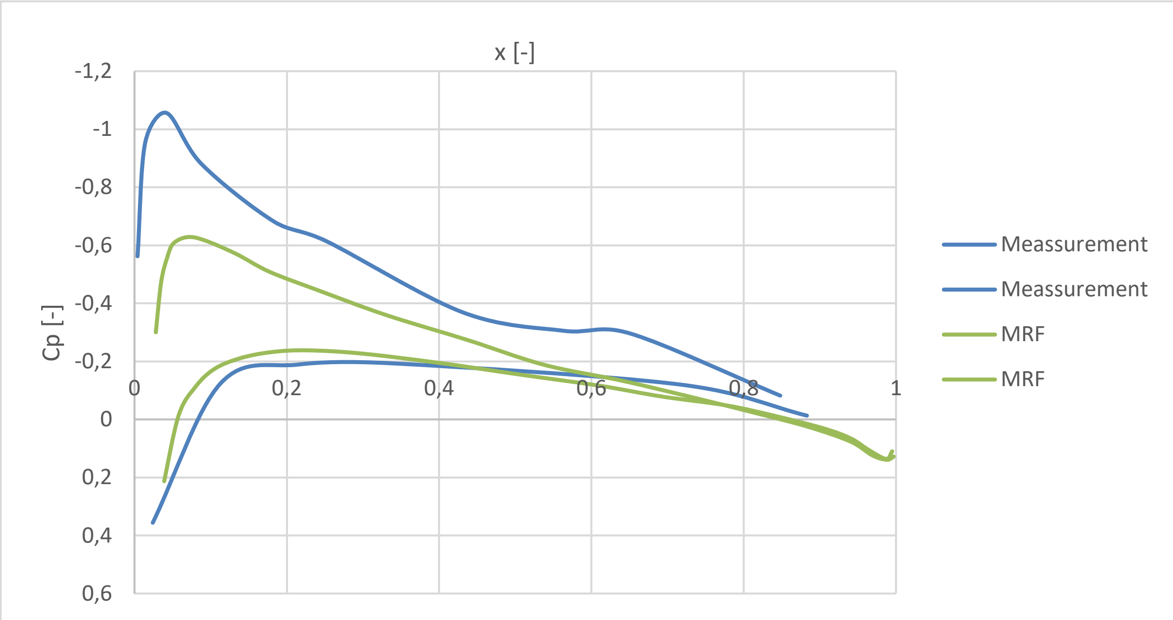

Figure 4 – Cp distribution

3. Conclusion and further work

A geometry was introduced for a two bladed fixed pitch UAV rotor. This geometry was then meshed using the Ansys Fluent Meshing software to provide simulation meshes for Ansys Fluent CFD computations. The meshes were different for two simulation methods, MRF and SM. The partial results showed that MRF computations are insufficient and will need to be reworked. Currently, computations for the SM method are ongoing. As these take a longer time, because of the higher cell count and also because the finer mesh requires a lower time-step to fulfil the CFL condition, they are not yet finished. Also, a rework of the MRF method is needed, because of a high degree of error in the results. One possible source of this is the mesh itself, which might be refined in the wrong areas or not refined enough. SM method results will show if the problem is with the mesh itself or with the accuracy of the method.

Next an investigation of transition trajectory will take place. Using a high-speed camera and flow visualisation, it will be experimentally measured how the inflow angle changes for normal UAV manoeuvres. Using this data, a user defined function (UDF) will be implemented into the pressure far field boundary condition, that will control the angle and velocity of the flow. Another UDF will control the rotor speed accordingly to the manoeuvre simulated. This will create a working test bed for acoustical CFD simulations, where an array of measuring points around the rotor will show how the noise changes transiently during the manoeuvre, not just with fixed inflow angle steps. As a final point, a new geometry of a rotor will be introduced, one that can be experimentally measured on an UAV and thus the results could be directly compared to real life measurements.

4. Contact Author Email Address

191407@vutbr.cz

5. Copyright Statement

6. Acknowledgement

This work has been supported by the project No. FSI-S-23-8163 funded by The Ministry of Education, Youth and Sport (MEYS, MŠMT in Czech) institutional support.

References

[1] HAVAS Prague. *Drone delivery from gadget to mainstream,* accessed 25th September 2024, <https://havas.cz/en/h-commerce-en/drone-delivery-from-gadget-to-mainstream/>

[2] Zhong S, Zhou P, Chen W, Jiang H, Wu H, Zhang X. An investigation of rotor aeroacoustics with unsteady motions and uncertainty factors. *Journal of Fluid Mechanics,* Vol. 956, 2023. DOI: https://doi.org/10.1017/jfm.2022.1097

[3] Zhong S, Zhou P, Fattah R, Zhang X. A revisit of the tonal noise of small rotors. *Proceedings of the royal society A,* Vol. 476 (2244), 2020. DOI: https://doi.org/10.1098/rspa.2020.0491

[4] Hanson D, Parzych D. Theory for noise of propellers in angular inflow with parametric studies and experimental verification. *NASA Contractor Report 4499.* 1993.

[5] Gennaretti M, Bernardini G, Serafini J, Trainelli L, Rolando A, Scandroglio A,… Paolone E. Acoustic prediction of helicopter unsteady manoeuvres. *41st European Rotorcraft Forum,* Munich, 2015

[6] Brès G A, Brentner K S, Perez G, Jones H E. Maneuvering rotorcraft noise prediction. *Journal of Sound and Vibration*, Vol. 275(3-5), pp 719-738. 2004.

[7] Chen H N, Brentner K, Shirey J. An investigation of transient rotor noise in maneuvering flight. *11th AIAA/CEAS Aeroacoustics Conference,* Monterey CA, p.n. 2903, 2005.

[8] Brentner K S, Lopes L V, Chen H N, Horn J F. Near real-time simulation of rotorcraft acoustics and flight dynamics. *Journal of aircraft*, Vol. 42(2), pp 347-355. 2005.

[9] Jamaluddin N S, Celik A, Baskaran K, Rezgui D, Azarpeyvand M. Aerodynamic noise analysis of tilting rotor in edgewise flow conditions. *Journal of Sound and Vibration*, Vol. 582, p.n. 118423, 2024.

[10] Jamaluddin N S, Celik A, Baskaran K, Rezgui D, Azarpeyvand M. Aerodynamics and Aeroacoustics Characterisation of Isolated Rotor in Hover and Transition to Forward Flight. *AIAA Aviation 2021 Forum,* virtual event, 2021. DOI: https://doi.org/10.2514/6.2021-2311

[11] Zawodny N, Pettingill N. Acoustic wind tunnel measurements of a quadcopter in hover and forward flight conditions. *INTER-NOISE and NOISE-CON Congress and Conference Proceedings*, Vol. 258 (7), pp. 487-500, 2018.

[12] Caradonna F X, Tung C. Experimental and analytical studies of a model helicopter rotor in hover. *NASA Technical Memorandum 81232*, 1981.

[13] ANSYS. Theory guide, *ANSYS FLUENT 23.0*, 2023.