Journal of Wind Engineering & Industrial Aerodynamics 218 (2021) 104776

Contents lists available at ScienceDirect

Journal of Wind Engineering & Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia

Assessment of the methodology for the CFD simulation of the flight of a quadcopter UAV

C. Paz, E. Suárez *, C. Gil, J. Vence

CINTECX, Universidade de Vigo, Campus As Lagoas-Marcosende, 36310, Vigo, Spain

ARTICLE INFO

Keywords:
UAV
Quadcopter
Aerodynamics
CFD
MRF
Sliding mesh
Thrust
Ground effect

ABSTRACT

The computational fluid dynamics (CFD) simulation is gaining attraction in the development of modern unmanned aerial vehicles (UAV), but few research has been made on quadcopters and the characterisation of the flows generated by the propellers, which determine the thrust capacity. Therefore, the purpose of this study is to assess the performance in the 3D flow simulation of the most promising methods: multiple reference frames (MRF) and sliding meshes. Additionally, the effect of the ground proximity has been included. The results for a sole propeller revealed both models as equivalent with respect to the evaluation of the ground effect, even though a noticeable deviation was observed in the thrust quantification. In the case of quadcopters, the relative position between blades and frame was proved as a key factor. Thus, similar rates of thrust change were obtained when minimising the superposition of the blade over the body arms in the MRF case. However, the thrust magnitude differed at least 11% at any simulated position. Assuming this deviation, the significantly lower computational cost of the MRF turns this model into a very interesting option. Finally, the influence of the relative blade-to-blade position in the sliding simulation was also assessed.

1. Introduction

Over the last years, the use of unmanned aerial vehicles (UAVs) is becoming common in a wide range of military, industrial and civilian applications (Bäckman et al., 2018; Chiang et al., 2019; Estrada and Ndoma, 2019). The versatility of these vehicles comes from a varied morphology both in size and platform design, which makes them adaptable to the requirements of each situation. Whereas fixed-wing aircrafts are more suitable for long flights in open environments (Kontogiannis and Ekaterinaris, 2013; Bravo-Mosquera et al., 2017; Panagiotou et al., 2018), multicopters stand out for short duration tasks either in open or close spaces due to the simplicity, maneuverability and the capacity of performing a vertical take-off. However, many of these tasks involve the flight in the proximity of walls or the ground, such as the inspection of surfaces and the delivery of cargo, which represents a challenge for the stability of these vehicles due to the well-known ground effect.

The ground effect, according to the Blade Element theory, implies an increase of the apparent thrust experienced by the vehicle. As approaching the ground, the downwards flow generated by the blades gets more obstructed and the induced velocity is therefore reduced. This phenomenon has been well studied in the field of conventional helicopters, leading to mathematical expressions that quantify this thrust variation for a sole propeller. The most widely accepted model is the one developed by Cheeseman and Bennett (1955), which involves only the radius of the propeller and the distance to ground as parameters. Based on its simplicity, many other researchers used this model to compare their results (Johnson, 1980; Powers et al., 2013; Sharf et al., 2014) and also similar expressions were proposed (Hayden, 1976).

Nevertheless, these single propeller models do not account for design aspects that become very important in the development of the multicopters such as: the use of tapered airfoils, a wide range of rotational speeds, or even the number of rotors. The effect of blade geometry and the rotational speed was experimentally studied by Deters et al. (2017) for different commercial propellers. The results showed an increase of the dimensionless thrust with the rpm that, in some cases, remains stable from a certain speed around 5000 rpm on. With respect to the number of rotors, several experiments brought to light differences between the flow generated by a sole propeller and a multicopter, being the quadcopter the most evaluated design. In this regard, Bernard et al. (2017) carried out an experimental campaign using blades of radius 0.1524 m and a cross-shaped quadcopter with a distance of 550 mm between opposite

* Corresponding author.
E-mail address: suarez@uvigo.es (E. Suárez).

https://doi.org/10.1016/j.jweia.2021.104776
Received 18 June 2021; Received in revised form 7 September 2021; Accepted 8 September 2021
Available online 15 September 2021

rotor axes. The results for an isolated propeller show a fairly good agreement with classical models, especially the Cheeseman and Bennett's formula, being the ground effect noticeable at about $H/R < 2$. On the contrary, this effect is extended in the quadcopter case to $H/R < 4$. Similar conclusions were drawn by Conyers et al. (2018) in their analysis of a custom quadcopter using Gemfan 9x4.7 nylon propellers and testing several distances between rotors. The thrust curves for a single isolated propeller fit well with the theoretical model and only small differences due to the operation in a sensitive range of Reynolds number might be discerned. However, significant variations are observed in the quadcopter evaluation. In this case, a prominent thrust increase is revealed at about $H/R = 4$, which is subsequently reduced to reach a minimum in the range of $1.5 < H/R < 2$. This minimum is associated with the effect known as Vortex Ring State. Thus, the results of these studies point that the classical models do not accurately predict the ground effect on quadcopters. As an approximation, some authors propose the use of an equivalent radius based on the sum of the areas of the individual rotors (Bangura and Mahony, 2012) or a parameter based on specific experimental data (Danjun et al., 2015) to adapt the Cheeseman and Bennett's model, but its general applicability is yet to be proven. Likewise, Sánchez-Cuevas et al. (2017) proposed a model that combines a potential flow with the method of images to predict the ground effect on quadcopters, obtaining a good agreement with their own experimental results. This model is based on geometrical parameters and includes a body lift term with an empirical coefficient. More complex models based on algorithms that account for the presence of walls to evaluate the forces experience by the quadcopter have been also proposed (Gao et al., 2019; He and Leang, 2020; McKinnon and Schoellig, 2020).

As it can be seen, the evaluation of the aerodynamic characteristics associated to the ground effect in quadcopters is a challenging task due to the number of variables involved and the interaction between them. In this respect, other study techniques such as wind tunnel tests (Schiano et al., 2014; Tomic et al., 2016; Smeur et al., 2018) or numerical simulations have risen as promising alternatives to real scale experimental tests. More specifically, several examples of computational fluid dynamics (CFD) simulations have been found in the literature. In this regard, the main objective is the proper reproduction of the flow field generated by the propeller. To this purpose, different approaches have been used: Kang and Sun (2000) added source terms into the Navier-Stokes equations to represent the rotor blades in a 2D analysis; Raza et al. (2017) performed a LES simulation of a building wake and combined the results with a mathematical model of the control method of a typical quadrotor to evaluate its performance; and Kaya and Özcan (2013) employed the fan boundary condition to a zero thickness propeller. Even so, none of these approaches reproduce the 3D flow generated by the rotor, accounting for the specific geometry of the blades and the body frame. Instead of, these approaches assign to the flow previously obtained terms or impose a uniform downwards velocity. In this regard, the only two techniques that allow for the modelling of the actual 3D flow created by specific blades spinning at a certain speed are the use of multiple reference frames (MRF) and the use of sliding meshes. The former one is a steady modelling of the flow field by means of the assignment of the rotational component to the velocity vector of the fluid cells. Since the rotation of the propeller is imposed in the flow equations and no transient simulation is required, the associated computational cost is relatively low. This method has been applied by Stajuda et al. (2016) in the simulation of a single isolated propeller to evaluate the optimal dimensions of the computational domain around the blade. Kutty and Rajendran (2017) also used the MRF model in their simulation of a sole propeller and different free-stream velocities, obtaining a good agreement with experimental results, especially at low advance ratios. On the other hand, the generation of relative velocity between domains using a sliding condition and non-conformal interfaces is a common practice in CFD simulations. In contrast to the MRF, this method requires the update of the mesh at each time-step and a transient solution, which increase considerably the calculation time and cost. In the field of aerodynamics, the studies of Zhang et al. (2020) and Joo et al. (2015) employed this method to evaluate the performance of the blades on wind turbines. However, the application of sliding meshes to quadcopters is less usual in the literature. Even so, Misiorowski et al. (2019) used it in their evaluation of the flow generated by a quadcopter in edgewise flight in both the cross and plus configurations. A velocity of 10 m/s and a pitch attitude of 5 deg were considered, and neither the drone body nor the arms were included. The results revealed a decrease in the generated lift in the cross configuration, especially in the backward rotors, whereas this lift is generally increased in the plus one. In addition, the validation of the simulation of a sole propeller against experimental values showed a good agreement in the thrust measurement.

Based on this revision of the state of the art in the field of CFD simulations of quadcopters, the objective of this paper is to compare and contrast the available methodologies that allow for the 3D modelling of the flow generated by a realistic propeller: MRF and sliding mesh. The main concerns are to evaluate how each method reproduces the flow field and the interaction of the four generated flows, and to quantify the thrust exerted on the quadcopter. Moreover, the effect of the ground proximity is also determined for both methodologies, simulating a wide range of distances from the quadcopter to the ground. Generally speaking, a sliding mesh calculation is considered to be more robust and yield more meaningful results, but the computational cost is significantly more elevated compared to the MRF (ANSYS, 2016). Therefore, it would be a matter of interest to evaluate the performance of both models under the specific conditions of a quadcopter, to measure the discrepancies in the thrust quantification, and to determine in which situations the MRF would be a more worthwhile option considering its lower computational cost. Thereby, the paper is divided in four sections. After this introduction contained in Section 1, in Section 2 the geometrical models of both the quadcopter body and the propellers are described; the computational methodologies and the boundary conditions of every performed simulation are presented; and the used grids are detailed. Next, the main results of the investigation are shown and discussed in Section 3. Finally, the conclusions of the research are summarised in Section 4.

2. Materials and methods

2.1. Geometrical model

The geometrical model selected for this study is a simplified version of the DJI Phantom 3 (DJI-Team, 2021). The original quadcopter consists of a cross-shaped body with four rotors and includes a camera, a camera support assembly and two support legs (see Fig. 1 (a)). Previous simulations have proven certain effect of these accessories on the thrust measurements. Therefore, the lower part of the body (camera, camera support and support legs) has been neglected in the current study to avoid considering its influence on the results (see Fig. 1 (b)).

Regarding the propellers, a sketch is shown in Fig. 1 (c), and the evolution of both the chord and the twist angle along the radius (centre to tip) is described in Fig. 1 (d). These dimensions correspond to the profile of the original blade geometry.

2.2. Numerical methodology and boundary conditions

In this study, different combinations of simulation settings have been employed to evaluate separately the effect of each of the following factors in the reproduction of the flow field and the quantification of the thrust:

- Geometrical model: sole propeller, quadcopter.
- Rotation methodology: MRF, sliding mesh.
- Distance to ground: based on H/R.

These settings are outlined in Fig. 2 and explained throughout this section.

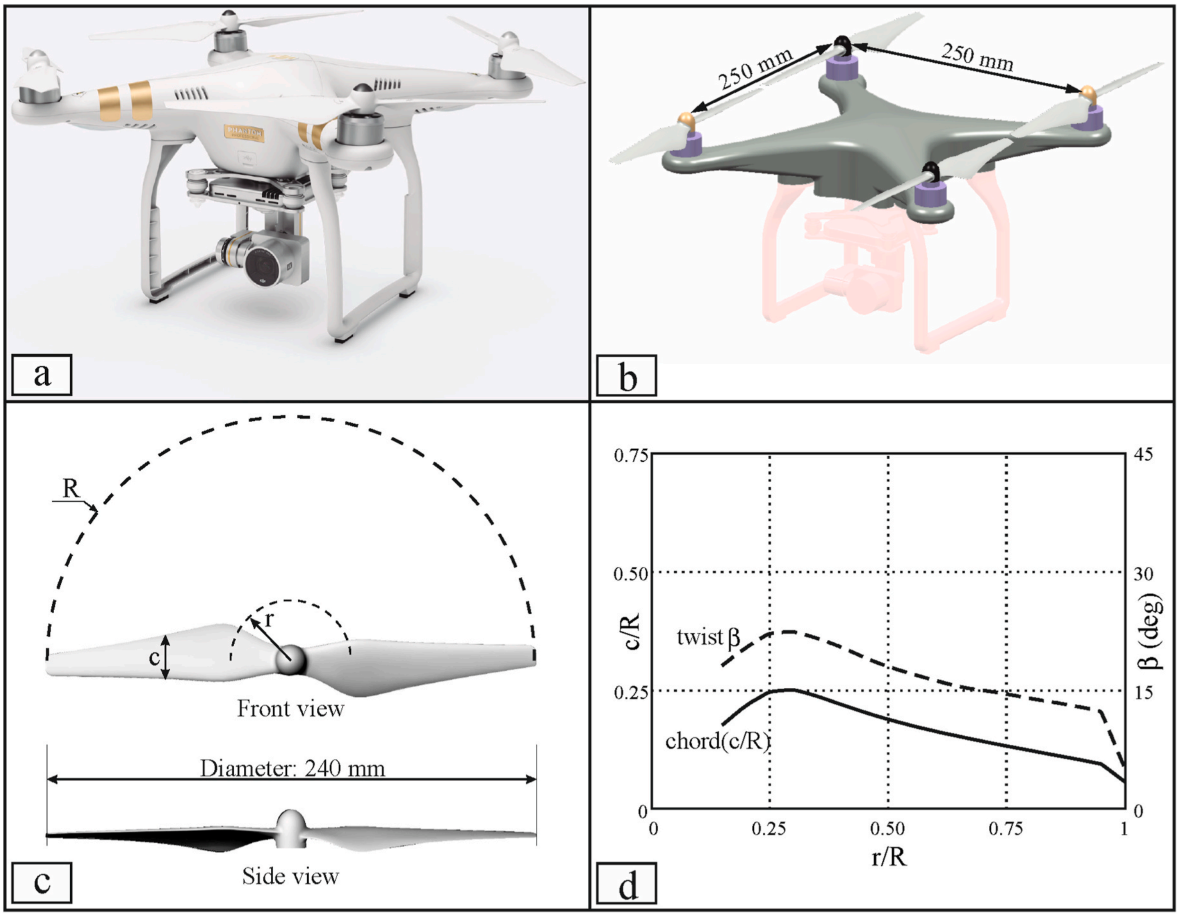

Fig. 1. DJI Phantom 3 geometry: (a) original, (b) CAD model, (c) propeller sketch, (d) propeller dimensions.

Firstly, the strategy shown in Fig. 2(a) was used to evaluate the performance of both the MRF and the sliding methodology in the reproduction of the flow generated by the rotation of a sole propeller. This situation had been previously analysed by computational methods (Paz et al., 2020) as well as through theoretical models (Cheeseman and Bennett, 1955; Hayden, 1976), which allows for the validation of the numerical process and a broader comparison of both methodologies and their relationship with the ground effect.

These rotation methodologies were subsequently assessed using the simplified model of the DJI Phantom 3 quadcopter (Fig. 2(b) and (c)). Both models shared the same geometrical model, computational domain, mesh and boundary conditions, being the only difference the method used to reproduce the flow generated by the blades. In contrast with a sole propeller (Fig. 2(a)), the combination of four rotors spinning at the same time represents a totally different situation in which the flow generated by each propeller is influenced by the presence of the others. In addition, this flow is not steady and constant during the entire cycle of rotation, but shows significant differences depending on the relative position of the propeller with respect to the drone arm. Furthermore, it is hypothesised that the relative position of each propeller with respect to the other propellers would also have a noticeable influence on the thrust experienced by the quadcopter.

Regarding the MRF model, the cylindrical region labelled as "MRF domain" (see Fig. 2(b)) contains the surfaces of the blade that are moving in the reality and that are responsible of generating the vortex tube that impulses the drone. However, this method considers a static region (and so a steady blade) and models the rotation by assigning a rotational component, which is function of the distance to the rotation axis, to the velocity vector of each surface cell. Thus, the effect of the rotational velocity is transmitted to the entire MRF domain and also to the global fluid domain through the pair of interfaces that allows the passing of fluid, reproducing the instantaneous flow actually generated. Hence, the MRF is often referred to as the "frozen rotor approach" (ANSYS, 2016). Since the walls inside the domain are static, the position of each blade with respect to the body arms and with respect to each other is significant. To evaluate this influence, 5 different models were simulated (MRF – A to MRF – E). The models A, C and D correspond to specific positions in which the four propellers have the same relative position with respect to the arm. On the contrary, in cases B and E, each pair of opposite propellers is orthogonal to the other. The nature of this method allows for the simulations to be performed in a steady way, reducing the computational requirements.

On the other hand, the method based on a "Sliding domain" is shown in Fig. 2(c). In this case, a sliding condition is applied to the interfaces that separate the cylindrical region containing the propeller, and the remaining fluid. This way, the mesh of the sliding domain is actually rotating at the specified speed with respect to the global domain and, subsequently, so the walls of the blades move. Therefore, the velocity vector of the fluid cells is modified due to the presence of the solid walls of the propellers, reproducing realistically the creation of the downwards flow. In contrast to the MRF model, this sliding method requires a transient simulation, being the movement of the mesh calculated using the rotation velocity and the time-step. The criterion to select this time-step was to guarantee that the blade does not cross more than a cell in each step, keeping a Courant number below 1. Thus, the time-step was dependent on the propeller radius and the mesh size.

All the strategies were also used to evaluate the effect of the ground proximity on both the sole propeller and the drone flight. For this purpose, the distance from the propeller/drone to the ground was progressively reduced. This allows for the evaluation of the flow behaviour