Numerical studies on modeling the near- and far-field wake vortex of a quadrotor in forward flight

Nathanael, Joshua Christian; Wang, John Chung-Hung; Low, Kin Huat

2021

Nathanael, J. C., Wang, J. C. & Low, K. H. (2021). 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.

<https://dx.doi.org/10.1177/09544100211029074>

<https://hdl.handle.net/10356/155247>

<https://doi.org/10.1177/09544100211029074>

© 2021 IMechE. All rights reserved. This paper was published in Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering and is made available with permission of IMechE.

Downloaded on 26 Apr 2023 18:41:25 SGT

Numerical Studies on Modeling the Near- and Far-Field Wake Vortex of a Quadrotor in Forward Flight

Journal Title
XX(X):1–19
©The Author(s) 2021
Reprints and permission:
sagepub.co.uk/journalsPermissions.nav
DOI: 10.1177/ToBeAssigned
www.sagepub.com/

SAGE

Joshua Christian Nathanael¹, C. H. John Wang¹, and Kin Huat Low²

Abstract

The operation of Unmanned Aerial Systems (UAS) is expected to keep increasing in the future with its rapid advancement and ever-expanding applications. One of the safety measures that need to be put into place for safe UAS operations is separation requirements. This paper presents the results from flow simulations of quadrotor propellers using the Overset method in ANSYS Fluent 19.2 and the Virtual Blade Model (VBM) method in OpenFOAM as part of ongoing work to establish a wake-vortex-based safe separation distance for UAS operations. The near-field flow simulation using Reynolds-averaged Navier-Stokes (RANS) turbulence model was validated with mesh convergence and turbulence model studies and verified with simulation and experimental data in the literature to ensure that the propeller simulation methods could generate a realistic near-field flow of a multi-rotor UAS in forward-flight. The far-field flow simulation using Large Eddy Simulation (LES) turbulence model was validated with a mesh convergence study by calculating and comparing the circulation strength of the vortices in the far-field flow of each mesh setup, but not verified due to a lack of existing experimental data.

Keywords

CFD, Quadrotor, Multi-rotor UAS, Wake turbulence, Separation minima

Introduction

Unmanned Aerial Systems (UAS) have undergone rapid advancement and ever-expanding applications in recent years. Many aspects of the UAS, from the controls for following set trajectories¹, privacy protection², positioning and navigation³, to battery charging⁴, are active topics in the current research pertaining to UAS. Numerous missions in various fields have also been experimented with UAS, from payload delivery⁵, road traffic monitoring⁶, construction inspection⁷, to survey of plantations⁸ and animals⁹. With the continuous development and increasing use of UAS, it is expected that more and more UAS will occupy the airspace in the near future, and likely that many of them will be operated remotely beyond visual line of sight (BVLOS)¹⁰.

Therefore, to prevent a UAS from negatively affecting other UAS while flying in the airspace, safety measures, such as separation requirements for example, need to be put in place. At the same time, these separation requirements should also be considered in terms of the effect on the resulting capacity of the airspace, as overly conservative requirements would decrease the overall utilization of the airspace and thus limiting the number

of UAS operations^{11,12}. So in formulating safe separation requirements, there is a balance to keep in mind between ensuring all UAS are flying safely and maximizing the airspace capacity for UAS operations.

Separation requirements have been long-established and applied for manned aircraft¹³, and the factors that contribute to such requirements, such as collision avoidance, navigational uncertainties, and wake turbulence effects, are also applicable for UAS operations. Several studies have been conducted to calculate the risk of collision on an unmanned aircraft to ensure safe operation alongside other aircraft, manned or unmanned, in the airspace^{14–17}. Additional study has also begun to investigate the navigation performance of common multi-rotor UAS¹⁸ which can be used to determine the separation requirement that takes

¹Air Traffic Management Research Institute (ATMRI), Nanyang Technological University, Singapore 637460

²School of Mechanical and Aerospace Engineering, Nanyang Technological University, Singapore 639798

Corresponding author:

Kin Huat Low, School of Mechanical and Aerospace Engineering, Nanyang Technological University, Singapore 639798

Email: mkhlow@ntu.edu.sg

the navigational errors into account. Furthermore, there are studies that look into the effect on a fixed-wing UAS under the influence of wake turbulence from a similar fixed-wing UAS^{19–21} or a larger manned aircraft²².

Although there have been numerous studies on the wake flow generated by rotary wing UAS, to the knowledge of the authors, so far there has been no study in the literature that investigates the wake field and flow dynamics at sufficiently large enough follow distance to be useful for wake-based separation discussions. Computational Fluid Dynamics (CFD) simulation of flow from a single rotating propeller model has been used to calculate and predict the thrust produced from the propeller, and several studies have shown the predicted thrust to be in agreement with experimental results in hover condition^{23–26}. In addition, both methods of flow simulation^{24,26,27} and experimental work^{28–30} have shown that, compared with a single isolated propeller, multiple propellers individually produce reduced thrust when they are rotating simultaneously in close proximity due to the flow interaction and interference among the propellers.

Flow simulation with CFD has also been used to give insight into the complex and detailed flow field from multiple propellers of UAS, whether with smaller propellers (Radius < 0.5 m) in hover^{24,26,31,32} and forward-flight^{33–36} conditions, or with larger propellers (Radius = 2 to 4 m) for studies on Urban Air Mobility (UAM) aircraft^{27,37–40}. Such insight could be helpful in understanding, for instance, how in forward-flight the thrust generation of one propeller is affected by the flow from the other propellers and thus changing the overall lift and moment on the vehicle^{33–35,38,39}, or how the downwash flow field influences the spray of particles from the vehicle^{31,32,36}. The crucial part, however, is the verification of the numerical calculations from the CFD simulation to make sure that the flow structure predicted is producible in an experimental setup^{31,36,41}. Nevertheless, most of these studies focus on the immediate wake structure close to the UAS itself, rather than the developed flow in the farther distance that another UAS might encounter.

While the use of a propeller model for CFD simulation may produce accurate results, the computational cost to perform the calculations can be very expensive due to the fine meshing required to fully resolve the complex flow on the surfaces of the propellers. Therefore, to simplify the computation while still maintaining a level of accuracy, analytical models known as the blade-element models can be used to calculate the lift and drag on each segment (blade-element) of the propeller. For their calculations, the blade-element models require the aircraft velocity, propeller's

rotational speed, and the velocity of the induced flow which can be estimated using various models including momentum models, lifting-line models, and vortex models⁴².

Several studies have used the blade-element model together with the momentum model, or the Blade-Element Momentum (BEM) model, to study the aerodynamics of small-scale propellers. The BEM model works well in certain flight conditions where there is little to no flow separation, but the model becomes inaccurate when there are strong three-dimensional effects due to the propeller geometry and/or operating conditions⁴³. There are studies that proposed corrections to the BEM model so that the model can better predict the performance of propellers in conditions with low Reynolds number (of around 10^4 to 10^5 ⁴⁴) which is often the case for propellers of small UAS^{45,46}. The BEM model has since been incorporated into CFD simulation in some studies to examine the flow from small UAS^{47,48} as well as large helicopters^{49,50}.

For this paper, the flow simulations of quadrotor propellers were performed in two parts: first using the Reynolds-averaged Navier-Stokes (RANS) turbulence models for the near-field region close to the propellers, and second using the Large Eddy Simulation (LES) turbulence model for the far-field region away from the propellers. For the flow simulations in the near-field region, in addition to using the full propeller model with the Overset mesh in the commercial flow solver ANSYS Fluent, the Virtual Blade Model (VBM) which is based on the BEM model was also used in the open-source flow solver OpenFOAM for comparison as a possible alternative. The flow simulation work in this study is a part of an ongoing effort by the authors to establish a wake-vortex-based safe separation distance for UAS operations.

The outline of this paper after this introduction is as follows: Numerical Methods and Schemes that were set up for the flow simulations in ANSYS Fluent and OpenFOAM, Near-Field Wake Validation and Verification in which flow simulations with RANS turbulence models were performed on a single rotating propeller and two counter-rotating propellers to verify the resulting near-field flow with the available data in the literature, Far-Field Wake Validation in which flow simulations with LES turbulence model were performed using the near-field flow from multiple rotating propellers to develop vortical flow in the far-field region and the resulting circulation strength of the vortices were compared between different meshes for a convergence study, then finally Conclusion with the key findings as well as areas to be covered for the future work of this study.

Numerical Methods and Schemes

Flow solvers like ANSYS Fluent and OpenFOAM solve mass (Eq. 1) and momentum (Eq. 2) conservation equations (known as the Navier-Stokes equations) to calculate the resulting flow in the domain of interest for the given boundary conditions.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = S_m \quad (1)$$

$$\frac{\partial}{\partial t} (\rho \vec{u}) + \nabla \cdot (\rho \vec{u} \otimes \vec{u}) = -\nabla p + \nabla \cdot \bar{\tau} + \rho \vec{g} + \vec{F} \quad (2)$$

S_m in Equation 1 is the source term that can equal to zero unless there is mass addition from the dispersed second phase (such as from vaporization of liquid), and the terms $\rho \vec{g}$ and $\vec{F}$ in Equation 2 represent the gravitational body force and external body forces which can equal to zero depending on the case considered. The stress tensor $\bar{\tau}$ in Equation 2 is given by the equation below.

$$\bar{\tau} = \mu \left[(\nabla \vec{u} + \nabla \vec{u}^T) - \frac{2}{3} (\nabla \cdot \vec{u}) I \right] \quad (3)$$

Turbulent flow in nature is highly variable and unsteady, consisting of swirls known as eddies with a wide range of length and time scales. One way a flow solver handles the calculations for turbulent flow is by decomposing the instantaneous velocity and scalars for the Navier-Stokes equations into the mean (ensemble-averaged or time-averaged) component and the fluctuating component.

$$u_i = U_i + u'_i \quad (4) \quad p = P + p' \quad (5)$$

Once these variables are substituted into the conservation equations (Eq. 1 and 2), the equations can be averaged to give the Reynolds-averaged Navier-Stokes (RANS) equations. Equations 6 and 7 show the RANS equations in Cartesian tensor form for the case of incompressible flow where the density of the fluid is constant in space and time while assuming the source term, body forces, and external forces equal to zero.

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (6)$$

$$\begin{aligned} \frac{\partial}{\partial t} (\rho U_i) + \frac{\partial}{\partial x_j} (\rho U_i U_j) = & - \frac{\partial P}{\partial x_i} \\ & + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] \\ & + \frac{\partial}{\partial x_j} \left(-\rho \overline{u'_i u'_j} \right) \end{aligned} \quad (7)$$

The term $-\rho \overline{u'_i u'_j}$ in Equation 7 represents the Reynolds stresses which can be modeled with an equation (Eq. 8 for incompressible flow) that relates the Reynolds stresses to the mean velocity gradients by using the Boussinesq eddy viscosity hypothesis.

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (8)$$

μ_t is the turbulent viscosity (also known as the eddy viscosity) and k is the turbulence kinetic energy, and they require additional equation(s) from a turbulence model to complete the set of the equations, containing only the mean values of velocity and pressure, that is solvable to calculate the turbulent flow. There are various turbulence models that can be utilized to solve the RANS equations, and the suitability of each model varies depending on the flow case, e.g. external, free-shear flow versus internal, wall-bounded flow.

The most common turbulence models that have been used to simulate the flow from a rotating propeller include the Spalart-Allmaras^{33,49}, $k - \epsilon$ ^{23,31}, and $k - \omega$ ^{24,32,41,50} turbulence models. The Spalart-Allmaras turbulence model employs one additional transport equation to be able to compute the turbulent viscosity (μ_t) while ignoring the turbulence kinetic energy (k) when calculating the Reynolds stresses for solving the RANS equations. The $k - \epsilon$ and $k - \omega$ turbulence models use two additional transport equations to compute the turbulence kinetic energy (k) and dissipation rate (ϵ or ω , the latter refers to the dissipation rate per unit volume and time), and the two variables are then used to compute the turbulent viscosity (μ_t) so that the Reynolds stresses can be calculated to solve the RANS equations. The standard forms of the $k - \epsilon$ and $k - \omega$ turbulence models have also been modified to overcome their initial shortcomings, thus producing added turbulence models that can be used such as the Renormalization Group (RNG) $k - \epsilon$, Realizable $k - \epsilon$, Baseline (BSL) $k - \omega$ and Shear-Stress Transport (SST) $k - \omega$.

Although solving turbulent flows with RANS equations may be cheaper in terms of computational power and time consumption, the results from the calculations may be lacking in detail due to the averaging of the flow variables to solve the RANS equations. Another method called Large Eddy Simulation (LES) directly resolves the larger eddies in the flow while modeling only the eddies that are smaller than the size of the spacing in the computational grid, such that the resulting flow simulated from LES contains more details as the turbulent flow is resolved as much as possible depending

on the computational grid that is used. This is accomplished by employing a filter function and applying it to the flow field

$$\overline{\phi(\mathbf{x}, t)} = \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \phi(\mathbf{r}, \tau) G(\mathbf{x} - \mathbf{r}, t - \tau) d\tau d\mathbf{r} \quad (9)$$

with ϕ being the variable of the flow and G being the filter function that is associated with a cutoff length and time scale.

Applying the filter function to the Navier-Stokes equations (Eq. 1 and 2) will give the filtered mass and momentum conservation equations. Equations 10 and 11 are the filtered equations for incompressible flow.

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (10)$$

$$\begin{aligned} \frac{\partial}{\partial t} (\rho \bar{u}_i) + \frac{\partial}{\partial x_j} (\rho \bar{u}_i \bar{u}_j) = & - \frac{\partial \bar{p}}{\partial x_i} \\ & + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] \end{aligned} \quad (11)$$

The term $\overline{u_i u_j}$ can be split up and written as

$$\overline{u_i u_j} = \bar{u}_i \bar{u}_j + \tau_{ij} \quad (12)$$

where τ_{ij} is the sub-grid scale stress tensor that represents the interaction between all the eddies in the flow from the larger, resolved eddies to the smaller eddies that are filtered in the process. Substituting equation 12 into 11 gives the equation below.

$$\begin{aligned} \frac{\partial}{\partial t} (\rho \bar{u}_i) + \frac{\partial}{\partial x_j} (\rho \bar{u}_i \bar{u}_j) = & - \frac{\partial \bar{p}}{\partial x_i} \\ & + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] \\ & - \frac{\partial \tau_{ij}}{\partial x_j} \end{aligned} \quad (13)$$

A Sub-Grid Scale (SGS) model used in LES relates the stress tensor τ_{ij} with the eddy viscosity μ_t using the Boussinesq eddy viscosity hypothesis in a way similar to RANS turbulence models, in that an added process of computing the eddy viscosity is required to calculate the turbulent flow. Several SGS models that can be used for LES include the Smagorinsky-Lilly model, the Wall-Adapting Local Eddy-Viscosity (WALE) model, and the kinetic energy model.

Furthermore, due to the use of the turbulent viscosity model in both RANS and LES calculations, the two methods can also be carried out using a hybrid model for the same flow case, where some regions of the flow (e.g. the flow near a wall) are solved using RANS calculations and the

other regions (e.g. the large detached flow) are solved using LES. One of the several hybrid models that have been developed is called the Detached Eddy Simulation (DES), and this has been used in several studies that simulate the flow from propellers of smaller UAS^{26,34,35} and larger UAM aircraft^{27,37–40}.

To determine the wake-based separation requirements, either in distance or in time, identifying the dominating dissipation mechanism for the evolution of far-field wake is essential. Therefore, to study the flow from UAS propellers for this paper, a near-field flow simulation was first done to initialize the wake near the propellers using the RANS turbulence model. The simulation of the far-field wake was then carried out using the LES turbulence model instead of the RANS turbulence model used in the near-field simulation. This is due to the wake vortex dissipation being dependent on small-scale non-uniform separation from the interaction between the counter-rotating vortex pair^{51–53} that the RANS turbulence models were not designed to capture, thus leading to excessive and unrealistic dissipation of the rotating flow.

For calculating a flow case in a given fluid domain, ANSYS Fluent and OpenFOAM use the finite volume method which solves the transport equations to compute the flow variables for every control volume in the computational grid. The equations, which initially apply to a fluid domain that is continuous in space and time, are discretized with numerical schemes so that the equations govern how the flow variables in the fluid domain relate spatially (i.e. the flow variables in one control volume with the surrounding control volume) and temporally (i.e. the flow variables at different points in time). In addition to the spatial and temporal discretization schemes, flow solvers also employ an algorithm called the pressure-velocity coupling which dictates how the different transport equations are solved, whether in a manner that is segregated (i.e. every flow variable is calculated sequentially) or coupled (i.e. all of the flow variables are calculated simultaneously).

For the near-field flow simulation with RANS turbulence models, some of the schemes that are set in ANSYS Fluent were kept as with the default settings, such as the spatial discretization for the gradient (Least Squares Cell Based), pressure (Second Order), and momentum (Second Order Upwind). The spatial discretizations for the turbulence kinetic energy and dissipation rate were changed to Second Order Upwind, and the Coupled pressure-velocity coupling was used as this was the only pressure-velocity coupling that was compatible with the Overset mesh in ANSYS Fluent 19.2. Meanwhile, the near-field flow simulation with

the VBM in OpenFOAM uses the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) pressure-velocity coupling, and the Gauss linear numerical scheme which has second order accuracy by using the Green-Gauss cell-based gradient evaluation and linear central differencing. It should be noted that central differencing schemes for momentum, turbulence kinetic energy, and dissipation rate in ANSYS Fluent were not available for RANS simulation, so the Second Order Upwind scheme was used instead.

The far-field flow simulation was conducted separately from the near-field flow simulation to reduce the computational cost given the limitation in the available resources, and due to the incompatibility between the LES turbulence model with the Overset and VBM methods. The Overset method in ANSYS Fluent is compatible with only a few turbulence models and the VBM method in OpenFOAM is only applicable to steady-state cases, for which the LES simulation is not suited due to its unsteady nature.

For LES simulation, ANSYS Fluent has different default settings for LES and they were all kept for this study, such as the SIMPLE pressure-velocity coupling, the spatial discretization for the gradient (Least Squares Cell Based), pressure (Second Order), and momentum (Bounded Central Differencing). The Bounded Second Order Implicit transient formulation was recommended and used in ANSYS Fluent. The far-field flow simulation in OpenFOAM used the pressure-velocity coupling that is a combination of the SIMPLE algorithm with the Pressure Implicit with Splitting of Operators (PISO) algorithm known as PIMPLE. Unlike in ANSYS Fluent, the SIMPLE pressure-velocity coupling in OpenFOAM is only applicable to steady-state simulations while PISO is applicable to transient simulations, and hence the combination of the two algorithms. The Gauss linear numerical scheme was also used for the far-field flow simulation in OpenFOAM with the backward time scheme which is second order implicit for the transient formulation.

Near-Field Wake Validation and Verification

The validation and verification of the near-field wake simulation results will be limited to single and multi-rotor in hover condition^{54,55}. This is due to a limited amount of experimental and simulation data available for verification purposes. Namely, the single rotor results will be verified with the simulation study by Céspedes and Lopez²⁴. The single and multi-rotor results are further compared qualitatively with the PIV visualizations conducted by Ning et al. from Iowa State University²⁹. While the group also published some measurements for a single propeller

in forward-flight configuration⁵⁶, the results were only published as a contour plot and thus cannot be meaningfully compared with the simulation results. For the validation of the simulation results, the mesh convergence and turbulence model comparison studies will be performed. The validation and verification process will be conducted following the framework established in the “Guide for the Verification and Validation of Computational Fluid Dynamics Simulations” published by the American Institute of Aeronautics and Astronautics (AIAA)⁵⁷.

Single Rotor

The single rotor simulations were carried out following the conditions presented in the study by Céspedes and Lopez²⁴, thus using 0.888kg/m^3 for the air density and 1.803kg/m-s for the dynamic viscosity. The propeller was set to operate at 3000rpm. The propeller was modeled based on the 3D scan of a propeller model with 0.4m diameter, provided by Prof. Ali Türker Kutay of the Middle East Technical University whose research group documented the thrust-performance of the same propeller in an earlier study²⁸.

The flow simulations in this study were performed using two methods: the first used the Overset method in ANSYS Fluent 19.2⁵⁴, and the second used the VBM method in OpenFOAM⁵⁵. Of the two techniques used to model the propeller-driven flow, the Overset method takes a more straightforward approach by modeling and meshing the propeller directly into a rotating domain that rotates at the speed of the propeller’s rotation. The interpolation function inside the Overset toolbox helps to translate the freestream, or continuum, flow to the propeller centric reference frame, solve the induced flow field inside the propeller reference frame, then pass the flow data back into the freestream domain. The positioning of the rotation zone relative to the continuum zone is presented in Fig. 1. The settings for the pressure inlet and outlet boundary conditions shown in Fig. 1 were kept at the default values with zero gauge pressure to simulate hover conditions with no vertical or horizontal motion.

The Overset-based simulation requires a highly resolved boundary-layer (expressed in lower y^+ value) to model the propeller-driven flow and capture the flow separation on the blade, as well as a relatively small time-step size to prevent instability with the flow interpolation. This makes Overset-based simulation of a multi-rotor UAS the more computational resource-intensive method for simulating the propeller-driven flow, while the results produced are likely to be more physically accurate in comparison to the VBM

Figure 1. The simulation domain layout (a) and the mesh (b) used in the Overset simulation study. The densely meshed disk in (b) is the rotating zone. D is the propeller's diameter.

Table 1. Overset rotation zone: mesh convergence study

Mesh Setup	Number of Cells	y^+ Range
Coarse	156,371	4.4 - 98.1
Medium	816,873	0.1 - 5.9
Fine	1,626,013	0.015 - 3.21

Table 2. Overset continuum zone: mesh convergence study

Mesh Setup	Number of Cells
Coarse	279,000
Medium	558,000
Fine	837,000

method. The mesh convergence study to validate the Overset solver was carried out in two parts, the first would vary the mesh density of the rotation zone while using the coarse mesh density for the continuum zone; the second would use the medium density rotation zone with continuum zone of different mesh densities. The overview of the mesh setup is presented in Tables 1 and 2.

In comparison, the Virtual Blade Model (VBM) in OpenFOAM was implemented from the BEM model under a specific class of codes called “rotorDiskSource”, initially for modeling the downwash from a helicopter rotor⁵⁸. The VBM method emulates the propeller-induced flow field using the blade-element theory, which treats each radial segment of the propeller as a 2D airfoil with no cross-flow. The sectional blade profile, propeller pitch, and chord lengths for the VBM were extracted from the propeller's 3D scan model, with the sectional lift and drag coefficient evaluated using JAVAfoil. Under this assumption, the angle of attack (α) of the propeller segment could be calculated as the arc-tangent of the tangential velocity ($V_{tan} = r \times \Omega$, where Ω is

Figure 2. Rotor downwash simulated using varying mesh refinement in the rotation zone compared with literature data.

Figure 3. Rotor downwash simulated using varying mesh refinement in the continuum zone compared with literature data.

Table 3. VBM mesh setups for the convergence study

Mesh Setup	Cells	Unrefined Mesh Size (10^{-3}m)	Refinement Zone (10^{-3}m)
Baseline	782,947	$78.1 \times 71.4 \times 68.2$	$9.7 \times 8.9 \times 8.5$
Coarse 1	433,506	$94.3 \times 85.7 \times 85.7$	$11.8 \times 10.7 \times 10.7$
Coarse 2	303,364	$111.1 \times 101.7 \times 85.7$	$6.9 \times 6.4 \times 5.4$
Fine 1	2,448,619	$53.2 \times 48.4 \times 46.9$	$6.7 \times 6.1 \times 5.9$
Fine 2	5,580,792	$40.3 \times 36.6 \times 35.7$	$5.0 \times 4.6 \times 4.5$

the propeller's rotational speed) and the inflow velocity as observed from the propeller's reference frame. The blade-element theory would, however, ignore the finite wing effect, where the flow becomes three-dimensional near the propeller tip as the air flows from the high-pressure underside to the low-pressure topside. This 3D flow effect could be addressed in the VBM using a tip model which emulates the changes in α near the propeller blade tip, or by simply setting a tip factor which ignores the force contribution by any element within a certain threshold from the blade tip. The force contribution by the blade-elements in the simulation time step is then applied onto the flow behind the propeller as the body-force term in the Navier-Stokes equation.

As the propeller-induced flow is emulated with a model, the computational cost using VBM is significantly lower than the Overset method and the resulting body-force contribution is largely independent of the mesh density in the region where VBM is applied. The emulation of the propeller-induced flow, however, will not be able to accurately predict 3D flow separation on the propeller and should only be used with a steady-state flow such as the case in the standard RANS turbulence model. The VBM simulation domain was generated as a rectangular mesh of length $10D$, width $6D$, and height $12D$ with local refinement applied as a cylindrical region of diameter $2D$ below the rotor to better capture the downwash flow, with D being the propeller's diameter. The boundary conditions for the VBM simulation were set as pressure inlet and outlet with zero gauge pressure, similar to those in the Overset simulation. A mesh convergence study was performed on the continuum mesh to validate that the resulting propeller-driven flow could be well resolved. Table 3 shows the different mesh setups that were tested.

The results from the mesh convergence study are presented in this paper as visualizations and qualitative analysis. The former compares the downward velocity contour for a slice of the fluid domain along the propeller axis, and the latter compares the extracted velocity profile from the flow simulations for this study and the ones in the literature. The downward velocity contour plots for a propeller in hover are presented in Figs. 2, 3, and 4 and compared to the visualization with the same setup by

Figure 4. Downwash velocity contour plots sliced along the rotor axis for the mesh convergence study from simulation using VBM and SST $k - \omega$ turbulence model.

Céspedes and Lopez²⁴. The mesh convergence studies were carried out using the SST $k - \omega$ turbulence model, following the simulation setup from the literature.

Overall the velocity contour plots show that the results from the Overset method are visually similar to the results from the literature for the given flow condition, regardless of the mesh density of either zone. Some minor variations could have existed due to the different strategies in the mesh generation and how the flow simulation was carried out. On the other hand, the velocity contour plots from the VBM method show a much more dispersed core flow, with the high downward speed region terminating much closer to the propeller, though the effect appears to be independent of the mesh density as well. The convergence of the mesh refinement level is confirmed in the velocity profile plots

(a) Varying Rotor Zone

(b) Varying Continuum Zone

Figure 5. Plots of downwash velocity profile from Overset simulation at one rotor radius distance below the propeller ($Z = 1.0R$, where R is the rotor radius) for the mesh convergence study as compared with Ref. ²⁴.

presented in Figs. 5 and 6, where the extracted velocity profile does not change (with the exception of moving from Coarse to Medium density for the rotating zone in the Overset method) with the increase in mesh density. This suggests that the difference seen between the VBM results and the literature results is not due to insufficient mesh density that failed to capture the flow features, but could be due to other issues such as the turbulence modeling.

The second part of the solver validation process is the turbulence model study, from which we would gain confidence that the correct turbulence model was used for the mesh and flow condition setups. Three turbulence models were tested for the Overset method using the Fine Rotating zone and the Coarse Continuum zone: the SST $k - \omega$ model, the Standard $k - \epsilon$ model, and the Realizable $k - \epsilon$ model. The popular Spalart-Allmaras turbulence model was not used

Figure 6. Plot of downwash velocity profile from VBM simulation at one rotor radius distance below the propeller ($Z = 1.0R$, where R is the rotor radius) for the mesh convergence study as compared with Ref. ²⁴.

as it was not compatible with the Overset implementation in version 19.2 of ANSYS Fluent. On the other hand, the VBM method tested the SST $k - \omega$ model, the Standard $k - \epsilon$, and the Spalart-Allmaras turbulence models using the Fine I mesh setup. Note that the $k - \epsilon$ model is known for its weakness in modeling highly sheared flow, such as the case of a propeller-driven flow, and is only used as a comparison. The downward velocity contour plots are shown in Fig. 7 and Fig. 8.

Indeed, the Standard $k - \epsilon$ model shows the most dissipation of the propeller-driven flow. The difference between the Overset and VBM results could be due to the difference in the discretization method, numerical interpolation method, or the numerical gradient calculation method. Overall though, the SST $k - \omega$ model appears to be the closest to the literature results for the Overset method while the Spalart-Allmaras model appears to perform the best for the VBM setup. This is confirmed in the downward velocity profile plot in Fig. 9. Overall, the validation and verification tests showed that for simulating the flow field generated by a single rotating propeller, the medium density mesh with the SST $k - \omega$ model would be the most suitable for the Overset method, and the medium (Baseline) mesh density with the Spalart-Allmaras model would be the best for the VBM method.

Figure 7. Downwash velocity contour plots for the turbulence model study from simulation using Overset.

Figure 8. Downwash velocity contour plots for the turbulence model study from simulation using VBM.

(a) Overset Method

(b) VBM Method

Figure 9. Downwash velocity profile plots at one rotor radius distance below the propeller ($Z = 1.0R$, where R is the rotor radius) for the turbulence model study with (a) Overset method and (b) VBM method.

Twin Rotors

The next step for the verification process is to ensure that the inter-propeller interference flow is sufficiently captured by the simulation setup, and flow simulation of twin counter-rotating rotors in hover condition was carried out to accomplish this. The flow simulation for the twin rotors case used the suitable mesh resolution and turbulence model identified for the Overset and VBM methods in the verification process for a single rotor, and so the mesh resolution near the propellers and directly downstream in the continuum zone was similar as in the single rotor case.

The setup for the simulation was in accordance with the Particle Image Velocimetry (PIV) experiment conducted at Iowa State University²⁹, as shown in Fig. 10, that investigated the influence of inter-propeller spacing on the thrust production. The boundary conditions for the Overset and VBM simulation were set as pressure inlet and outlet with zero gauge pressure, similar to those in the single rotor simulations. Another propeller model that is more similar to the one used in the PIV experiment was created for the Overset simulation instead of using the previous propeller for the single rotor simulations. 4860RPM was used as the propeller rotational speed in the simulations to follow the rotational speed used in the PIV experiment.

Unfortunately, the PIV data were only presented in downward velocity contour and planar velocity glyph (that shows the flow rotation), and a detailed velocity profile could not be extracted from the literature with the tools available to us. Instead, the PIV visualization available was used as a qualitative comparison of the performance of the two simulation methods before we move on to the simulation of quadrotor propellers in forward-flight.

Fig. 11 shows the downward velocity contour plot from the PIV (top row) compared to the simulation results using the VBM method (middle row) and the Overset method (bottom row). The velocity profiles were extracted at $0.1D$ (left) and $1.0D$ (right) downstream from the propeller rotation plane, with D being the propeller's diameter of 0.24m. Note that the discontinuity shown in Fig. 11e is not due to actual discontinuity in flow results, but due to the difference in how the downward velocity was averaged between the rotating zone and the stationary continuum zone, and this difference became apparent in ParaView upon post-processing of the flow results combined from the two zones. The Overset method appears to capture the initial flow separation better (marked out in Fig. 11a) than the VBM method, while the VBM method shows a more realistic downstream flow pattern in comparison to the Overset method.

Figure 10. Schematic showing the propeller model and test-stand setup from the Iowa State PIV experiments. The schematic is extracted from Ref.²⁹.

The results are further compared in Fig. 12 with the velocity contour extracted along the propeller axis and parallel to the “ground” from the PIV setup. The downward velocity contour shows similar behavior among the three visualizations presented, although minor differences can be seen in the inter-propeller gap where the VBM failed to fully capture the low-pressure region. This is likely due to a combination of factors such as the cut-off style tip factor used in this implementation of VBM and insufficient distance from the VBM interface for the momentum injection to be fully developed (as shown in Fig. 12b). Overall though, both methods produced similar results as the PIV visualization. The difference could be caused by the uncertainty with the propeller model, as the model presented in the literature was under-defined thus requiring some guesswork to recreate for this study.

With no additional experimental data available for verification of quadrotor wake in forward-flight, we had to settle with proving first that the propeller models in this study create a downstream flow that matches the velocity profile from previous studies, and the rotational flow and the resulting interference qualitatively match the visualization from high-quality PIV experiments. In other words, we could accept with relative certainty that both propeller simulation methods were able to capture the flow generated by the propellers of a multi-rotor UAS as well as the inter-propeller interference, therefore they should generate realistic near-field wake simulation of a multi-rotor UAS in forward-flight.

Figure 11. Time-averaged downwash velocity contour at $Y = 0.1D$ (left) and $Y = 1.0D$ (right) from: (a), (b) PIV measurements from Ref. ²⁹; (c), (d) VBM; and (e), (f) Overset simulation. D is the diameter of the propeller.

(a) PIV Measurements

(b) VBM Simulation

(c) Overset Simulation

Figure 12. Time-averaged downwash velocity contour from (a) PIV measurements in Ref.²⁹, (b) the VBM, and (c) the Overset simulation.

Far-Field Wake Validation

The far-field wake simulation domain consisted of a rectangular domain that spanned from 1 – 11m behind the forward motor of the multi-rotor. A schematic showing the relative position of the far-field flow domain compared to the near-field setup is shown in Fig. 13. The near-field simulations were generated with the multi-rotor set to a 10° forward tilt, with a forward horizontal speed of 10m/s and propeller rotational speed of 4860RPM. The velocity wake profile from the near-field simulations was then extracted and mapped onto the front side of the far-field wake simulation domain to be the velocity inlet, while the velocity inlet on the upper side above maintained a uniformly same speed as the multi-rotor forward horizontal speed of 10m/s. The other sides of the far-field wake simulation on the left, right, rear, and below were set as pressure outlets with zero gauge pressure.

Due to the different structures of the near-field flow from the Overset method and the VBM method, the simulations using the LES turbulence model were conducted using the respective flow solvers, i.e. the LES simulation using the near-field results from the Overset method (the case of which will be called LES - Overset) was run using ANSYS Fluent and the LES simulation using the VBM generated near-field results (the case of which will be called LES - VBM) was run using OpenFOAM. The dynamic Smagorinsky-Lilly sub-grid scale model was used for the LES simulation of the far-field wake due to the high velocity gradient in the far-field flow that caused the Smagorinsky constant required for the calculation to be different across neighboring cells in the computational grid. By using the dynamic model, the Smagorinsky constant could be dynamically determined across the time and space of the far-field wake simulation instead of specifying it as a universal fixed value beforehand. Note that only the mesh convergence study was conducted for the far-field simulation. This is because the LES turbulence model for both solvers has been extensively tested with rotational/vortical flow^{59–62}, and the OpenFOAM solver has been previously used for the simulation of fixed-wing wake-vortex evolution near the ground^{63,64}.

The mesh convergence study was carried out for the mesh counts outlined in Table 4. Note that one of the Fluent simulations (labeled as “Adapted” in the table) was performed using a mesh adaption where more mesh points (finer discretization) were inserted into a previously used mesh within the high fluid-stress region to help in capturing the important flow features. While the Adaptive Mesh Refinement (AMR) method is not typically used in LES

Figure 13. The vertical sliced view of the $Z = 0$ m plane showing the near-field velocity contour (tilted square domain on the right) and the mesh used for the preliminary LES study (rectangular mesh on the left).

Table 4. Mesh counts used in the mesh convergence study

LES - Overset		LES - VBM	
Mesh Setup	Number of Cells	Mesh Setup	Number of Cells
Medium	5.7 million	Coarse	12 million
Fine	15.7 million	Baseline	23.5 million
Adapted	34.2 million	Fine	40 million

simulation as commutation error could emerge due to filter-grid size mismatch, such an error was not observed in this study. Also, note that the two simulation domains did not have the same buffer space around the region of interest needed to prevent boundary condition induced flow artifacts due to the different implementation in each solver; this means that the mesh counts between the two methods shown in the table are not directly comparable.

The circulation of the vortex pair at a specific follow distance behind the multi-rotor was computed by first identifying the vortex core positions of the left and right primary vortex. This could be accomplished by multiplying (or dividing) the vorticity ($\nabla \times \vec{U}$) with the pressure for the sample slice before finding the position with the minimum and maximum values. The circulation of that vortex could then be computed by taking a circular line integral of the velocity vector or surface integral of the vorticity vector at a desired radius around the vortex core.

For typical civil aviation applications, the averaged wake vortex strength is determined using the 5-15 m circulation measurements (Γ_{5-15}), which is a popular method for measuring wake vortex strength in civil aviation⁶⁵ that averages the circulation between radii $r = 5$ m to $r = 15$ m for an aircraft with vortex separation of $b_0 = 47.1$ m based

on a typical wingspan of commercial transport class aircraft, e.g. the Airbus A340. As the far-field wake in this study was generated from propellers with 0.24 m diameter, the vortex separation from a multi-rotor with similar propellers was taken as around $b'_0 = 0.5$ m (a little more than 2 times the propeller diameter). The average circulation was therefore performed by scaling down the measurement radius by a factor of $0.5/47.1$, and the circulation charts presented in Figs. 14 and 15 show the average circulation using the scaled-down radii between $r = 5$ cm to $r = 16$ cm at different follow distance behind the multi-rotor.

Time averaging was then performed on the circulation data to remove outliers. The mesh convergence study showed that as mesh resolution increases with the increasing mesh count (or with the additional refinement from the mesh adaption), the circulation measurement would increase until convergence is reached, at which point increasing mesh count would not significantly alter the circulation measurements. Based on the results, the “Fine” mesh for LES - Overset and “Baseline” mesh for LES - VBM would be sufficiently accurate for use in future studies.

A rough comparison of the initial vortex strength of fixed-wing aircraft could be estimated using $\Gamma_0 = W/s\rho Ub$, where W is the aircraft weight (or vertical component of the thrust for multi-rotor), s is the wing loading coefficient (typically $\pi/4$ for fixed-wing aircraft, ~ 0.9 for rotorcraft), ρ is the air density (1.225 kg/m^3 at sea level), U is the aircraft travel speed, and b is the wingspan. This would roughly place the expected initial vortex strength between 2.0 to $2.3 \text{ m}^2/\text{s}$, assuming no thrust is lost due to the propeller flow

(a) Left Vortex

(b) Right Vortex

Figure 14. Mesh convergence plot for far-field wake with initial condition generated by the Overset Method.

(a) Left Vortex

(b) Right Vortex

Figure 15. Mesh convergence plot for far-field wake with initial condition generated by the VBM method.

interference which could further reduce this value. Note that the formulation of Γ_0 was developed for fixed-wing aircraft, and is not directly applicable to rotorcraft, let alone multi-rotor UAS.

Although both LES - Overset and LES - VBM gave a similar value of the initial vortex strength, the vortex circulation charts in Figs. 14 and 15 show that the wake vortex in LES -VBM loses its strength much faster than the wake vortex in LES - Overset. The difference in the trend of the far-field vortex strength might be due to the different resulting far-field flow from each case, as shown by the far-field wake visualization in Fig. 16. The vortices in the LES - VBM far-field flow appeared to decompose into several vortices, one of which descended farther than the other, while the vortices in the LES - Overset wake structure didn't separate during their development.

Apart from the near-field velocity profile that was mapped for the front velocity inlet, all settings and boundary conditions for the LES simulation were kept similar between LES - Overset and LES - VBM. From the wake visualization of the near-field flow shown in Fig. 17, there appeared to be stronger downwash generated from the rear propeller in the VBM method such that the vortices contributed from the rear propellers became separate from the vortices produced by the front propellers, while in the Overset method the downwash from the rear propeller was not as strong and the vortices from the rear propeller would eventually add on to the other vortices from the front propeller.

This discrepancy might be due to the wake from the front propellers affecting the inflow for the rear propellers, the phenomenon of which might be handled differently between the Overset method and the VBM method in this study. Therefore, further work is required on generating the near-field wake of a multi-rotor using both the Overset and VBM methods to ensure that they agree on the resulting flow from a multi-rotor in forward-flight condition. Furthermore, due to the complexities of the far-field flow from a multi-rotor, the circulation measurement based on Γ_{5-15} might not be suitable to capture and represent the full impact of the wake vortex turbulence from the multi-rotor. Experimental work of visualizing and measuring the velocities of the wake flow from a multi-rotor in the forward-flight condition is also needed to verify the flow simulations and ensure that they are in agreement with the experimental setup.

Figure 16. Q-criterion isosurface ($Q = 20$) visualization from LES simulation using the near-field results from (a) the Overset method and (b) the VBM method.

Figure 17. Q-criterion isosurface ($Q = 20$) visualization of the near-field results from RANS simulation using (a) the Overset method and (b) the VBM method.

Conclusion

In the first part of this study, the near-field flow simulation of quadrotor propellers in hover was carried out using the RANS turbulence model in ANSYS Fluent with the Overset method and in OpenFOAM with the VBM method. The flow simulation methods have been validated with mesh convergence and turbulence model studies and verified with available simulation and experimental results in the literature. From validating and verifying the flow simulation of a single rotor in hover, the SST $k - \omega$ turbulence model was chosen for the Overset method, while the Spalart-Allmaras turbulence model was chosen for the VBM method.

From validating and verifying the flow simulation of twin counter-rotating rotors, it was observed that the Overset method appeared to capture the inter-propeller interference better than the VBM method, although the VBM method generated a more realistic downstream flow. Due to a lack of experimental data of the flow from multiple propellers in forward-flight, the validation and verification of the near-field flow in this study were accepted as enough confirmation

that the flow simulation methods should be able to generate realistic near-field wake of a multi-rotor UAS in forward-flight.

Following that, the two simulation methods were applied to generate the near-field wake of multiple propellers in forward-flight condition. The resulting near-field flow was further utilized for the far-field flow simulation using the LES turbulence model with the dynamic Smagorinsky-Lilly sub-grid scale model, which has been often demonstrated in previous studies as effective in simulating the wake-vortex flow evolution. The far-field flow simulation was validated through a mesh convergence study by calculating and comparing the circulation strength of the vortices in the far-field flow of each mesh setup.

While initially the vortex strength of the flow from the two simulation methods was similar, there was a difference in the dissipation of the vortex strength. This was possibly due to the discrepancy between the Overset and VBM methods in simulating the downwash flow from the rear propellers under the influence of the wake from the front

propellers in the forward-flight condition, thus resulting in the different near-field velocity profile used to generate the far-field flow. Further work is therefore still required on generating the near-field wake of multiple propellers in forward-flight condition using the Overset and VBM methods. Furthermore, the far-field flow could not be verified due to a lack of existing experimental data.

For the future works of this study, there are several areas that should be investigated to improve our confidence in the wake simulation results. First is by creating a more realistic simulation setup to predict a more accurate flow from multi-rotor UAS, and this can be done by including the body structure of the multi-rotor UAS and putting in realistic UAS trim condition (i.e. the cruising speed, the corresponding attitude angles, and the individual rotor rotational speeds) for the near-field flow simulation. Second is by running experimental studies, such as LIDAR or PIV (both subjected to availability) to provide experimental data with which flow simulations of multi-rotor UAS can be verified, or through UAS flight response measurements to study the effect of wake turbulence from multi-rotor UAS.

Furthermore, building up from the UAS wake flow studies using flow simulation, the far-field flow data can be coupled with the software-in-the-loop UAS flight simulation for the UAS wake vortex encounter study⁵⁵. The flight response of the follower UAS can be studied for various following distances behind the leader UAS to determine the appropriate separation distance between UAS. In the end, a recommendation on the safe separation distance between UAS can be implemented for UAS traffic management (UTM) to ensure the safety of UAS during their operations⁶⁶.

Acknowledgements

This research is supported by the Civil Aviation Authority of Singapore and the Nanyang Technological University, Singapore under their collaboration in the Air Traffic Management Research Institute. Any opinions, findings, conclusions, or recommendations expressed in this material are those of the authors and do not reflect the views of the Civil Aviation Authority of Singapore.

Declaration of conflicting interests

The authors declare that there is no conflict of interest.

References

1. Koksai N, An H and Fidan B. Backstepping-based adaptive control of a quadrotor UAV with guaranteed tracking performance. *ISA Transactions* 2020; 105: 98–110. DOI: 10.1016/j.isatra.2020.06.006.
2. Deebak B and Al-Turjman F. A smart lightweight privacy preservation scheme for IoT-based UAV communication systems. *Computer Communications* 2020; 162(April): 102–117. DOI:10.1016/j.comcom.2020.08.016.
3. da Penha Neto G, de Campos Velho HF and Shiguemori EH. UAV Autonomous Navigation by Image Processing with Uncertainty Trajectory Estimation. In *Lecture Notes in Mechanical Engineering*, volume 1. Springer International Publishing. ISBN 9783030536688, 2021. pp. 211–221. DOI: 10.1007/978-3-030-53669-5_16.
4. Xie K, Xu J and Pan Z. Research and application of anti-offset wireless charging plant protection UAV. *Electrical Engineering* 2020; 102(4): 2529–2537. DOI:10.1007/s00202-020-01049-9.
5. Raj R and Murray C. The multiple flying sidekicks traveling salesman problem with variable drone speeds. *Transportation Research Part C: Emerging Technologies* 2020; 120(September): 102813. DOI:10.1016/j.trc.2020.102813.
6. Outay F, Mengash HA and Adnan M. Applications of unmanned aerial vehicle (UAV) in road safety, traffic and highway infrastructure management: Recent advances and challenges. *Transportation Research Part A: Policy and Practice* 2020; 141(July): 116–129. DOI:10.1016/j.tra.2020.09.018.
7. Debus P and Rodehorst V. Multi-scale Flight Path Planning for UAS Building Inspection. In *Lecture Notes in Civil Engineering*, volume 98. Springer International Publishing. ISBN 9783030512958, 2021. pp. 1069–1085. DOI:10.1007/978-3-030-51295-8_74.
8. Ferreira MP, de Almeida DRA, Papa DdA et al. Individual tree detection and species classification of Amazonian palms using UAV images and deep learning. *Forest Ecology and Management* 2020; 475(April): 118397. DOI:10.1016/j.foreco.2020.118397.
9. Fiori L, Martinez E, Orams MB et al. Using Unmanned Aerial Vehicles (UAVs) to assess humpback whale behavioral responses to swim-with interactions in Vava'u, Kingdom of Tonga. *Journal of Sustainable Tourism* 2020; 28(11): 1743–1761. DOI:10.1080/09669582.2020.1758706.
10. Konert A and Kasprzyk P. Drones Are Flying outside of Segregated Airspace in Poland. *Journal of Intelligent & Robotic Systems* 2020; 100(2): 483–491. DOI:10.1007/s10846-019-01145-4.
11. Bulusu V and Polishchuk V. A threshold based airspace capacity estimation method for UAS traffic management. In *2017 Annual IEEE International Systems Conference (SysCon)*. IEEE. ISBN 978-1-5090-4623-2, pp. 1–7. DOI: 10.1109/SYSCON.2017.7934758.

12. Tan DY, Chi W, Bin Mohamed Salleh MF et al. Study on impact of separation distance to traffic management for small UAS operations in urban environment. *Advances in Transdisciplinary Engineering* 2017; 5: 39–46. DOI:10.3233/978-1-61499-779-5-39.
13. International Civil Aviation Organization (ICAO). *Procedures for Air Navigation Services - Air Traffic Management - PANS-ATM Doc 4444*. November, 2016. ISBN 978-92-9258-081-0.
14. Weibel R, Edwards M and Fernandes C. Establishing a Risk-Based Separation Standard for Unmanned Aircraft Self Separation. In *11th AIAA Aviation Technology, Integration, and Operations (ATIO) Conference*. June, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-60086-941-9, pp. 14–17. DOI:10.2514/6.2011-6921.
15. McFadyen A and Martin T. Understanding Vertical Collision Risk and Navigation Performance for Unmanned Aircraft. In *2018 IEEE/AIAA 37th Digital Avionics Systems Conference (DASC)*, volume 2018-Sept. IEEE. ISBN 978-1-5386-4112-5, pp. 1–10. DOI:10.1109/DASC.2018.8569707.
16. Weinert A, Campbell S, Vela A et al. Well-Clear Recommendation for Small Unmanned Aircraft Systems Based on Unmitigated Collision Risk. *Journal of Air Transportation* 2018; 26(3): 113–122. DOI:10.2514/1.D0091.
17. McFadyen A. Probabilistic Determination of Maximum Safe Altitudes for Unmanned Traffic Management. In *2019 IEEE/AIAA 38th Digital Avionics Systems Conference (DASC)*, volume 2019-Sept. IEEE. ISBN 978-1-7281-0649-6, pp. 1–10. DOI:10.1109/DASC43569.2019.9081741.
18. Kallinen V, Martin T and McFadyen A. Required Navigation Performance Specifications for Unmanned Aircraft Based on UTM Flight Trials. In *2020 International Conference on Unmanned Aircraft Systems (ICUAS)*. IEEE. ISBN 978-1-7281-4278-4, pp. 196–203. DOI:10.1109/ICUAS48674.2020.9213980.
19. Saban D, Whidborne JF and Cooke AK. Simulation of wake vortex effects for UAVs in close formation flight. *The Aeronautical Journal* 2009; 113(1149): 727–738. DOI:10.1017/S0001924000003389.
20. Tian P, He A, Chao H et al. Wake Encounter Simulation and Flight Validation with UAV Close Formation Flight. In *AIAA Guidance, Navigation, and Control Conference*. January, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-450-3, pp. 1–12. DOI: 10.2514/6.2017-1910.
21. Kazarin P and Golubev V. New results in wake vortex evolution, interaction and control. In *AIP Conference Proceedings*, volume 2046. ISBN 9780735417724, p. 020033. DOI:10.1063/1.5081553.
22. Mi Bg and Zhan H. Numerical Simulation of the Static and Dynamic Aerodynamics of a UAV under Wake Flows. *Journal of Advanced Transportation* 2019; 2019: 1–12. DOI: 10.1155/2019/6326794.
23. Kutty H and Rajendran P. 3D CFD Simulation and Experimental Validation of Small APC Slow Flyer Propeller Blade. *Aerospace* 2017; 4(1): 10. DOI:10.3390/aerospace4010010.
24. Céspedes JF and Lopez OD. Simulation and validation of the aerodynamic performance of a quadcopter in hover condition using overset mesh. In *AIAA Aviation 2019 Forum*. June, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-589-0, pp. 1–14. DOI: 10.2514/6.2019-2824.
25. Pérez Gordillo A, Villegas Santos J, Lopez Mejia O et al. Numerical and Experimental Estimation of the Efficiency of a Quadcopter Rotor Operating at Hover. *Energies* 2019; 12(2): 261. DOI:10.3390/en12020261.
26. Thai AD, Jain R and Grace SM. CFD Validation of Small Quadrotor Performance using CREATE™ - AV Helios. *The Vertical Flight Society - Forum 75: The Future of Vertical Flight - Proceedings of the 75th Annual Forum and Technology Display* 2019; (May 13–16, 2019).
27. Yoon S, Lee HC and Pulliam TH. Computational Analysis of Multi-Rotor Flows. In *54th AIAA Aerospace Sciences Meeting, AIAA SciTech Forum*, volume 0. Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-393-3. DOI:10.2514/6.2016-0812.
28. Kaya D, Kutay AT and Tekinalp O. Experimental Investigation of Optimal Gap Distance between Rotors of a Quadrotor UAV. In *AIAA Atmospheric Flight Mechanics Conference*. June, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-493-0, pp. 1–9. DOI:10.2514/6.2017-3894.
29. Zhou W, Ning Z, Li H et al. An Experimental Investigation on Rotor-to-Rotor Interactions of Small UAV Propellers. In *35th AIAA Applied Aerodynamics Conference*. June, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-501-2, pp. 1–16. DOI: 10.2514/6.2017-3744.
30. Shukla D and Komerath N. Multirotor Drone Aerodynamic Interaction Investigation. *Drones* 2018; 2(4): 43. DOI:10.3390/drones2040043.
31. Fengbo Y, Xinyu X, Ling Z et al. Numerical simulation and experimental verification on downwash air flow of six-rotor agricultural unmanned aerial vehicle in hover. *International Journal of Agricultural and Biological Engineering* 2017; 10(4): 41–53. DOI:10.25165/j.ijabe.20171004.3077.

32. Zheng Y, Yang S, Liu X et al. The computational fluid dynamic modeling of downwash flow field for a six-rotor UAV. *Frontiers of Agricultural Science and Engineering* 2018; 5(2): 159–167. DOI:10.15302/J-FASE-2018216.
33. Hwang JY, Jung MK and Kwon OJ. Numerical Study of Aerodynamic Performance of a Multirotor Unmanned-Aerial-Vehicle Configuration. *Journal of Aircraft* 2015; 52(3): 839–846. DOI:10.2514/1.C032828.
34. Misiorowski M, Gandhi F and Oberai AA. Computational Study on Rotor Interactional Effects for a Quadcopter in Edgewise Flight. *AIAA Journal* 2019; 57(12): 5309–5319. DOI:10.2514/1.J058369.
35. Ventura Diaz P and Yoon S. High-Fidelity Computational Aerodynamics of Multi-Rotor Unmanned Aerial Vehicles. In *2018 AIAA Aerospace Sciences Meeting*. 210059, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-524-1, pp. 1–22. DOI:10.2514/6.2018-1266.
36. Wen S, Han J, Ning Z et al. Numerical analysis and validation of spray distributions disturbed by quad-rotor drone wake at different flight speeds. *Computers and Electronics in Agriculture* 2019; 166(September): 105036. DOI:10.1016/j.compag.2019.105036.
37. Jia Z and Lee S. Acoustic analysis of a quadrotor eVTOL design via high-fidelity simulations. In *25th AIAA/CEAS Aeroacoustics Conference, 2019*. May, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 9781624105883, pp. 1–24. DOI:10.2514/6.2019-2631.
38. Diaz PV, Johnson W, Ahmad J et al. Computational study of the side-by-side urban air taxi concept. In *The Vertical Flight Society - Forum 75: The Future of Vertical Flight - Proceedings of the 75th Annual Forum and Technology Display*.
39. Diaz PV and Yoon S. Computational study of NASA's quadrotor urban air taxi concept. In *AIAA Scitech 2020 Forum*, volume 1 PartF. Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 9781624105951. DOI:10.2514/6.2020-0302.
40. Sagaga J and Lee S. CFD Hover Predictions for the Side-by-Side Urban Air Taxi Concept Rotor. In *AIAA AVIATION 2020 FORUM*, volume 1 PartF. Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-598-2, pp. 1–15. DOI:10.2514/6.2020-2795.
41. Cerny M and Breitsamter C. Investigation of small-scale propellers under non-axial inflow conditions. *Aerospace Science and Technology* 2020; 106: 106048. DOI:10.1016/j.ast.2020.106048.
42. Gur O and Rosen A. Comparison between blade-element models of propellers. *The Aeronautical Journal* 2008; 112(1138): 689–704. DOI:10.1017/S0001924000002669.
43. Carroll J and Marcum D. Comparison of a Blade Element Momentum Model to 3D CFD Simulations for Small Scale Propellers. *SAE International Journal of Aerospace* 2013; 6(2): 2013-01-2270. DOI:10.4271/2013-01-2270.
44. Winslow J, Otsuka H, Govindarajan B et al. Basic Understanding of Airfoil Characteristics at Low Reynolds Numbers (104–105). *Journal of Aircraft* 2018; 55(3): 1050–1061. DOI:10.2514/1.C034415.
45. McCrink M and Gregory JW. Blade Element Momentum Modeling of Low- Re Small UAS Electric Propulsion Systems. In *33rd AIAA Applied Aerodynamics Conference*. June, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-363-6, pp. 1–23. DOI: 10.2514/6.2015-3296.
46. MacNeill R and Verstraete D. Blade element momentum theory extended to model low Reynolds number propeller performance. *The Aeronautical Journal* 2017; 121(1240): 835–857. DOI:10.1017/aer.2017.32.
47. Chiew JJ and Aftosmis MJ. Efficient Simulation of Multi-rotor Configurations with Low Reynolds Number Propellers. In *2018 Applied Aerodynamics Conference*. Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-559-3. DOI:10.2514/6.2018-4119.
48. Stremel PM and Ahmad J. Overflow Analysis of Unmanned Aircraft Systems. In *AIAA Scitech 2019 Forum*. January, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-578-4, pp. 1–14. DOI: 10.2514/6.2019-1337.
49. Kusunov A, Kusunov S, Mikhailov S et al. Main rotor-body action for virtual blades model. *EPJ Web of Conferences* 2018; 180: 02050. DOI:10.1051/epjconf/201818002050.
50. Barakos G, Fitzgibbon T, Kusunov A et al. CFD simulation of helicopter rotor flow based on unsteady actuator disk model. *Chinese Journal of Aeronautics* 2020; 33(9): 2313–2328. DOI: 10.1016/j.cja.2020.03.021.
51. Holzäpfel F, Gerz T and Baumann R. The turbulent decay of trailing vortex pairs in stably stratified environments. *Aerospace Science and Technology* 2001; 5(2): 95–108. DOI: 10.1016/S1270-9638(00)01090-7.
52. Breitsamter C. Wake vortex characteristics of transport aircraft. *Progress in Aerospace Sciences* 2011; 47(2): 89–134. DOI: 10.1016/j.paerosci.2010.09.002.
53. Leweke T, Le Dizès S and Williamson CH. Dynamics and Instabilities of Vortex Pairs. *Annual Review of Fluid Mechanics* 2016; 48(1): 507–541. DOI:10.1146/annurev-fluid-122414-034558.
54. Nathanael JC, Wang CHJ and Low KH. Preliminary Investigation of Wake Vortex Generated by Spinning Quadrotor Propellers Using Overset Mesh. In *AIAA Scitech 2021 Forum*.

- January, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-609-5, pp. 1–13. DOI: 10.2514/6.2021-1309.
55. Wang CHJ, Nathanael JC, Ng EM et al. Framework for the Estimation of Safe Wake Separation Distance between Same-Track Multi-Rotor UAS. In *AIAA Scitech 2021 Forum*. January, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-609-5, pp. 1–14. DOI: 10.2514/6.2021-0708.
 56. Ning Z, Wlezien RW and Hu H. An Experimental Study on Small UAV Propellers with Serrated Trailing Edges. In *47th AIAA Fluid Dynamics Conference*. June, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-500-5, pp. 1–17. DOI:10.2514/6.2017-3813.
 57. Computational Fluid Dynamics Committee. *Guide for the Verification and Validation of Computational Fluid Dynamics Simulations (AIAA G-077-1998(2002))*. Washington, DC: American Institute of Aeronautics and Astronautics, Inc., 1998. ISBN 978-1-56347-285-5. DOI:10.2514/4.472855.
 58. Wahono S. Openfoam script learning: Development of Virtual Blade Model for Modelling Helicopter Rotor Downwash in OpenFOAM. Technical report, Defense Science and Technology Organization (Australia), 2013.
 59. Mo JO, Choudhry A, Arjomandi M et al. Effects of wind speed changes on wake instability of a wind turbine in a virtual wind tunnel using large eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics* 2013; 117: 38–56. DOI:10.1016/j.jweia.2013.03.007.
 60. Vlček P, Kysela B, Jirout T et al. Large eddy simulation of a pitched blade impeller mixed vessel – Comparison with LDA measurements. *Chemical Engineering Research and Design* 2016; 108(1998): 42–48. DOI:10.1016/j.cherd.2016.02.020.
 61. Tellez-Alvarez J, Strijhak S, Kharchi R et al. 3D Numerical Simulation of Wind Turbines and Fractal Dimension Analysis. In *2018 International Conference on Wind Energy and Applications in Algeria (ICWEAA)*. IEEE. ISBN 978-1-5386-6933-4, pp. 1–5. DOI:10.1109/ICWEAA.2018.8605052.
 62. Kamliya Jawahar H, Lin Y and Savill M. Large eddy simulation of airfoil self-noise using OpenFOAM. *Aircraft Engineering and Aerospace Technology* 2018; 90(1): 126–133. DOI:10.1108/AEAT-05-2015-0130.
 63. Wang CJ, Zhao D, Schlüter J et al. LES study on the shape effect of ground obstacles on wake vortex dissipation. *Aerospace Science and Technology* 2017; 63: 245–258. DOI: 10.1016/j.ast.2016.12.032.
 64. Wang CHJ, Zhao D, Schlüter J et al. Computational Investigation of Varying Plate-Line Geometry and Placement on Wake Vortex Dissipation. *Journal of Aircraft* 2018; 55(1): 133–144. DOI:10.2514/1.C034346.
 65. Ahmad NN, Proctor FH, Limon Duparcmeur FM et al. Review of Idealized Aircraft Wake Vortex Models. In *52nd Aerospace Sciences Meeting*. Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-256-1, pp. 1–28. DOI:10.2514/6.2014-0927.
 66. Mohamed Salleh MFB, Wanchao C, Wang Z et al. Preliminary Concept of Adaptive Urban Airspace Management for Unmanned Aircraft Operations. In *2018 AIAA Information Systems-AIAA Infotech @ Aerospace*. 209989, Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN 978-1-62410-527-2, pp. 1–12. DOI: 10.2514/6.2018-2260.