Washington University in St. Louis

Washington University Open Scholarship

Mechanical Engineering and Materials Science
Independent Study

Mechanical Engineering & Materials Science

4-30-2018

Conceptual and Numerical Analysis of Active Wingtip Vortex Cancellation in Propeller-Driven Electric Aircraft

Peter D. Sharpe

Washington University in St. Louis

Ramesh K. Agarwal

Washington University in St. Louis

Follow this and additional works at: <https://openscholarship.wustl.edu/mems500>

Recommended Citation

Sharpe, Peter D. and Agarwal, Ramesh K., "Conceptual and Numerical Analysis of Active Wingtip Vortex Cancellation in Propeller-Driven Electric Aircraft" (2018). *Mechanical Engineering and Materials Science Independent Study*. 62.

<https://openscholarship.wustl.edu/mems500/62>

This Final Report is brought to you for free and open access by the Mechanical Engineering & Materials Science at Washington University Open Scholarship. It has been accepted for inclusion in Mechanical Engineering and Materials Science Independent Study by an authorized administrator of Washington University Open Scholarship. For more information, please contact digital@wumail.wustl.edu.

Conceptual and Numerical Analysis of Active Wingtip Vortex Cancellation in Propeller-Driven Electric Aircraft

Peter Sharpe* and Ramesh K. Agarwal†
Washington University in St. Louis, St. Louis, MO 63130

As battery and electric motor technology continues to advance rapidly, propeller-driven electric aircraft are likely to become a significant part of the aviation market in the near future. One proposed design configuration for electric aircraft involves using large, wingtip-mounted propellers to actively cancel wingtip vortices, a method called active wingtip vortex cancellation (AWVC). By reclaiming part of the kinetic energy that would otherwise be lost to tip vortex formation, drag is decreased. In addition, the induced spanwise flow and upwash from the propeller causes the spanwise lift distribution to remain more uniform at the wingtips, increasing lift.

Previous wind tunnel testing of this configuration characterized a significant increase in lift and decrease in drag, particularly in low-aspect-ratio configurations. This paper builds on that research by examining several test cases with a 3D, transient, viscous, sliding mesh CFD analysis in an effort to validate numerical methods for future conceptual design studies. In addition, many practical considerations regarding the implementation of this design are analyzed.

Geometry from the aforementioned wind tunnel literature was reconstructed and analyzed. CFD indicated an 18.1% increase in lift and 5.1% increase in net thrust was possible solely through the phenomenon of AWVC. Furthermore, this CFD analysis matched wind tunnel data to within approximately 1%, validating the CFD approach for the analysis of more exotic configurations involving active wingtip vortex cancellation.

Nomenclature

α	= Angle of attack	C_L	= Lift coefficient
$\alpha_{L=0}$	= Zero-lift angle of attack	CFD	= Computational fluid dynamics
β	= Sideslip angle, nose-left positive	e	= Span efficiency factor
β_p	= Propeller geometric twist angle	M	= Local Mach number
ρ	= Density	r	= Radial position along the propeller blade
AR	= Aspect ratio ($\frac{b^2}{S}$)	r_{max}	= Propeller tip radius
AWVC	= Active Wingtip Vortex Cancellation	S	= Projected wing planform area
b	= Wingspan	$\vec{u}$	= Velocity
C_D	= Drag coefficient	u_∞	= Freestream velocity magnitude
$C_{D,i}$	= Induced drag coefficient	w	= Downwash

I. Introduction

A. Overview

All airplanes have wings of finite span, which necessitates the consideration of spanwise flow effects when analyzing the aerodynamics of finite wings. The most fundamental of these spanwise flow effects is the wingtip vortex. These vortices form at the tips of any lifting wing, and they are a direct consequence of the fact that the pressure distribution around a lifting wing tends towards ambient at the wingtips. This generates a spanwise pressure gradient above and below the wing; for a wing generating positive lift, this gradient induces an outboard acceleration on fluid elements below the wing and an inboard acceleration for those above the wing. This difference in spanwise velocity results in streamwise vorticity, which is most clearly visualized in the form of two bound vortices at the wingtips.

*Undergraduate Researcher, Department of Mechanical Engineering and Materials Science, AIAA Student Member

†William Palm Professor of Engineering, Department of Mechanical Engineering and Materials Science, AIAA Member

Under classical potential flow assumptions, Prandtl's lifting line theory can be used to analyze this phenomenon [1]. Using this theory, flow over a lifting wing is modeled as a superposition of infinite horseshoe vortices of infinitesimal strength, in accordance with the Helmholtz vortex theorems for inviscid, incompressible flow. This vortex system acts to induce a downwash along the wing, which can be calculated using the Biot-Savart law. This downwash causes the local "incoming" velocity vector $\vec{u}$ to tilt downwards, such that $\vec{u} = \vec{u}_\infty + w\hat{k}_w$, where $\hat{i}_w$, $\hat{j}_w$, and $\hat{k}_w$ denote the unit vectors of the standard wind axes coordinate system. This leads to two negative effects:

- 1) The effective angle of attack, α , over any wing section is decreased. This leads to decreased lift.
- 2) The local lift vector begins to tilt "backwards", since lift is produced perpendicular to the local incoming flow (and thus strictly upwards in the case of an induced downwash). This leads to increased drag in the form of induced drag.

Because of these effects, the L/D ratio of a finite wing is always lower than that of its theoretical infinite counterpart. These two negative effects are magnified as aspect ratio decreases, as shown in the respective relations: [1]

$$1) C_{D,i} = \frac{C_L^2}{\pi e AR}.$$

$$2) C_L = (\alpha - \alpha_{L=0})(a_0/[1 + (a_0/[\pi AR])(1 + \tau)]), \text{ where } a_0 \text{ and } \tau \text{ are effectively constants. } (a_0 = \frac{dC_L}{d\alpha} \text{ for the 2D airfoil section; } \tau \text{ is a function of the wing's Fourier coefficients } A_n \text{ from the analytical circulation derivation [1])}$$

In order to mitigate these effects, most aircraft designed for aerodynamic efficiency use relatively large aspect ratios (most transport aircraft have aspect ratios on the order of 10, while some high-efficiency sailplanes can have aspect ratios exceeding 30) [2].

However, high aspect ratio configurations come with drawbacks as well; most notably, structural effects (weight, strength, stiffness/aeroelasticity, etc.) enforce a practical limit on aspect ratio. Thus, as with almost all other decisions during the conceptual aircraft design process, aspect ratio selection is a trade-off. This paper analyzes an idea that could tip this trade-off towards low-aspect ratio designs, particularly in the realm of the small, propeller-driven, electric aircraft that have become popular in recent years.

B. Conceptual Design

The idea in question was first proposed by aerodynamicist Charles H. Zimmerman in the 1930s, at what would later become NASA Langley Research Center. Zimmerman proposed a discoidal aircraft with large, counter-rotating propellers mounted at the wingtips [3]. These propellers rotated in the inboard-up direction, which was quite unconventional at the time. The idea behind Zimmerman's design was to use the streamwise vorticity induced by the propellers to actively cancel the wingtip vortices, minimizing the detrimental spanwise flow effects that are typical of low-aspect-ratio designs.

Aircraft manufacturer Vought took it upon themselves to build a flying prototype, entitled the V-173. This prototype, illustrated in Fig. 1, flew nearly 200 times. According to test pilots, the reduction in induced drag and increase in lift caused by the propellers resulted in superb low-speed flying qualities [4].

Testing continued through the late 1940s with an all-metal version, the Vought XF5U. However, the program was canceled due to the waning military relevance of propeller aircraft in the jet age, the difficulties associated with transferring mechanical power out to the wingtips (the 1,350-hp motors required to drive the massive propellers needed to be mounted farther inboard due to the thin wingtips), the massive landing gear that the large propellers required in order to maintain ground clearance, and program cost and time overruns [4].

Because of this, any possibility of active wingtip vortex cancellation was essentially nullified for nearly 60 years as propeller propulsion was largely replaced by the turbofan. However, propeller-driven aircraft have received renewed interest in recent years, as huge leaps in battery and electric motor technology have finally made electric aircraft viable. In particular, electric propulsion already dominates the small Unmanned Aerial Vehicle (UAV) and Micro Air Vehicle (MAV) niche, due to the weight associated with piston engines and the impracticality of efficient turbojets at such a small scale.

Thus, the configuration that is proposed in this paper is this: **A low-aspect-ratio conventional small UAV with large wingtip-mounted propellers spinning inboard-up.** This design with inboard-up propellers will be referred to

Fig. 1 The Vought V-173 "Flying Pancake" in flight [5]

as the "proposed configuration" or "active wingtip vortex cancellation" (AWVC) configuration. Throughout this paper, this design will be contrasted with an identical design with the propellers spinning outboard-up (the "conventional configuration"). The reason that the proposed configuration is measured against the conventional configuration, as opposed to the clean configuration (no propeller), is to isolate the active-wingtip-vortex-cancellation effects in question from the simple prop-wash effects that are already known to exist.

C. Proposed Benefits

This configuration has the potential to bring the following benefits to electric UAVs:

- 1) By increasing the lift towards the wingtips, the overall wing loading is increased. This means that less wing area is required to generate an equal amount of lift, so the wetted area can be reduced. This reduces skin-friction drag, which is a major factor at the low Reynolds numbers of UAVs and MAVs where viscous effects and boundary layers become more important.
- 2) Assuming the aircraft has ailerons in the prop wash near the wingtips, the configuration naturally has blown surfaces. Blown surfaces are a form of thrust vectoring caused by having control surfaces in the wake of a propulsion system. By deflecting these control surfaces, a torque can be generated from the prop-wash (even when $u_\infty = 0$). By combining the left and right aileron, rolling moments can be generated. Assuming the aerodynamic center of the aileron is aft of the center of gravity, as is typically the case, a pitching moment can be created as well. Finally, by providing differential power to the left and right motors, yaw control can be gained as well. Thus, the aircraft has full three-axis control in all flight conditions, potentially allowing post-stall maneuvering or even hovering flight, with the help of a flight controller and sufficient thrust.
- 3) If wing area is held as a constant and a wingspan is to be selected, the implementation of active wingtip vortex cancellation will tend to "push" the design trade towards shorter wingspans. For constant area, this results in longer wing chords and higher Reynolds numbers at the wing. At the low Reynolds number of UAVs and MAVs (approximately $Re = 1E5$), the maximum L/D of a wing is very Reynolds-sensitive due to the onset of turbulent flow, as illustrated in Fig. 2. In other words, $d(L/D_{max})/d(Re)$ is very high near the baseline flight regime. Thus, a design trade that results in longer chords is very beneficial aerodynamically.

Fig. 2 Reynolds-sensitivity of L/D_{max} for a generic NACA 2412 airfoil; generated with XFOIL [6]

- 4) The increased chord of the wing also results in a thicker wing; these two effects allow for useful payload within the wing, as is the case with the V173 and XF5U.
- 5) Extremely wide flight envelopes been demonstrated with this configuration with regard to speed; for example, the Vought XF5U had a maximum speed of nearly 550 mph and a stall speed of just 40 mph [4], a ratio of nearly 14. This lends itself to STOL (Short TakeOff and Landing) operations and use in unimproved airfields.
- 6) Maneuverability is greatly improved as aspect ratio decreases. This is because moment of inertia scales with distance to the rotational axis squared, while control moments are typically scale linearly with the distance to the rotational axis. Thus, an airplane with its mass pulled in towards the center typically achieves much higher angular rates (particularly in roll), all else being equal.

- 7) Because of the decreased wing area (due to increased wing loading), the UAV will have lower observability; this could be of particular relevance for reducing visual clutter in civilian logistics operations or for military applications.

This configuration also addresses many of the weaknesses of the XF5U and other similar previous configurations:

- 1) Electric propulsion overcomes the mechanical complexity of the XF5U, simply because its complex mechanical transmission can be replaced by mounting a brushless electric motor directly at the wingtip. Electric motors are typically much smaller than combustion engines of equivalent power, and the smaller propulsion system interferes less with structural members.
- 2) UAV use potentially overcomes the long landing gear that were a weakness of the XF5U. Without a human pilot, and with a much lighter platform, there are many options. The airplane could be hand- or air-launched, while relying on folding props that feather upon landing. Another option is to use cheap, disposable landing gear. Alternatively, the aircraft could perform Vertical-TakeOff-and-Landing (VTOL), as it has full three-axis control.
- 3) Traditionally, outboard-up rotation has been more common than inboard-up because of reliability: in the event of a single engine failure, outboard-up typically results in more stable lateral modes. This is because the roll moment from prop torque partially cancels the roll moment from asymmetric-thrust-induced sideslip (given $dl/d\beta < 0$, where l denotes the roll moment and β denotes the sideslip angle). However, absolute reliability is less critical for pilotless vehicles, and engine-out capability is much less of a concern due to the greatly increased reliability of electric engines [7]. It is likely that the performance benefits characterized in the following section would vastly outweigh this minor reliability penalty.

II. Numerical Analysis

In order to validate and characterize the concept of Active Wingtip Vortex Cancellation (AWVC), a numerical analysis was performed.

A. Method Selection

Both potential flow and Navier-Stokes methods were considered for this analysis. Generally, potential flow methods have many advantages in conceptual design analysis: potential flow solutions have been shown to be robust and fairly accurate for many external aerodynamics problems, while having orders of magnitude less computational cost than Navier-Stokes methods. However, potential flow methods have some unfortunate drawbacks that render them less desirable for this particular problem:

- Classical potential flow methods such as lifting line theory or its more common extension, the vortex lattice method, excel in the analysis of medium- to high-aspect ratio wings. However, their accuracy begins to break down when analyzing low-aspect ratio configurations. While there are methods that address this (most notably, the 3D panel method), these methods have difficulty modeling multiple disconnected surfaces in close proximity (i.e. a wing and propeller), due to the propensity of the upwind wake to convect through the downstream surface [8].
- Propeller modeling is of lower fidelity in potential flow methods. The actuator-disk model induces no streamwise vorticity, which is critical to AWVC. It is possible to approximate vorticity using a rotating-actuator-disk model based on blade-element-theory, but neither of the two potential flow solvers used in the lab (XFLR5 and VSPAero) support this when combined with 3D panel method wing modeling.
- The inviscid assumption of potential flow theory starts to break down as Reynolds number decreases to small UAV scales. Additionally, while viscous effects might be less prominent near the propeller's tip, viscosity dominates the inner regions where the vortex core responsible for AWVC is formed. The irrotationality assumption is also violated because of the important rotational effects in the propeller wake's vortex core.
- A potential flow solution to this problem was already attempted in 2012 by Dimchev at TU Delft [9]. While overall lift calculations were generally accurate, the potential flow analysis that was presented appears to underestimate the lift difference between the inboard-up and outboard-up cases by approximately a factor of 2 when compared to wind-tunnel data (Fig. 5.15 in [9]).

Because of the deficiencies of a potential flow solution, a Navier-Stokes method was selected.

B. Geometry

The geometry was created in a combination of OpenVSP (Open Vehicle Sketch Pad, developed at NASA Ames) and ANSYS DesignModeler. The geometry was matched as closely as possible to that of the Dimchev experiment [9], in an

effort to later validate the numerical results against their wind-tunnel data.

The geometry consists of a straight, untapered, and untwisted wing with a NACA 642-015 airfoil section. The wing has a half-span of 0.325 m and a chord of 0.240 m, corresponding to $AR = 2.71$. There is a nacelle mounted at the wingtip (that is, the centerline of the nacelle is coincident with the nominal half-span) consisting of a half-ellipsoid, cylinder, and secant ogive, from front to back. The cylindrical section of the nacelle has a diameter of 0.07 m, and extends longitudinally from the airfoil's trailing edge to 0.180 m forward of the airfoil's leading edge. The half-ellipsoid extends 0.050 m forward of the cylinder, while the ogive extends 0.150 m aft of the cylinder. This is depicted in Fig. 3.

Fig. 3 Fixed components of the test geometry, all dimensions in meters.

The propeller was also modeled using dimensions from the Dimchev paper, which resulted in a propeller of the following characteristics: The four-bladed propeller had a constant Clark Y airfoil from root to tip. The propeller blade angle β_p measured at $3/4$ of r_{max} was 23° , and the propeller diameter was 23.6 cm (radius $r_{max} = 0.118$ m). Propeller rake and skew were zero. The propeller chord and twist were fixed at several points along the radius of the blade, shown in Table 1; these points were then interpolated using a piecewise cubic hermite interpolating polynomial (PCHIP) spline. The propeller was placed such that the midpoint of the propeller's root chord-line was 0.202 m forward of the wing's leading edge. Two versions of the geometry were created; one with inboard-up-oriented propeller blades, and the other with outboard-up propeller blades.

Table 1 Propeller chord (c/r_{max}) and twist angle (β_p), as a function of radial location (r/r_{max})

r/r_{max}	c/r_{max}	β_p (deg)
0.23482	0.09607	40.30
0.35224	0.12414	34.25
0.46964	0.15182	30.00
0.58706	0.16448	26.30
0.70446	0.16045	23.45
0.82188	0.14674	21.30
0.91972	0.12717	19.80
1.00000	0.11111	19.80

Because of the interaction between rotating and fixed objects in the flowfield, the sliding mesh model was selected (this will be discussed in more detail later). This model required the creation of an axisymmetric rotating region within the geometry itself. This rotating region was a cylinder with its axis coincident to the rotation axis of the propeller. The cylinder had a diameter of 0.280 m; this was slightly larger than the propeller diameter of 0.236 m in an effort to allow the propeller tip vortex to convect solely through the rotating domain's downstream face (thereby reducing numerical diffusion caused by large flow gradients shearing across the interface). The cylinder extended 0.015 m forward and aft of the plane of the propeller (the plane normal to the propeller axis and coincident with the midpoint of the propeller's root chord-line). A key feature to note is the intersection of the rotating region with the nacelle; this created a small region on the nacelle's surface that rotated with the propellers like a prop hub, in an effort to relieve boundary layer discontinuities from the rapidly-spinning propeller blades.

The fluid domain was a rectangular prism extending 3 m from the wing's root leading edge in every direction except aft; the domain was extended to 4 m from the body in the aft direction to more accurately capture the turbulent wake. A symmetry condition about the wing's root face was desired in order to reduce computational cost; thus, the entire domain was cut in half.

It was chosen to simulate a case at $\alpha = 10^\circ$. This number was selected because it represented a moderate lifting

condition that was known to be pre-stall, according to wind tunnel data from Dimchev [9]. The typical approach to model nonzero angles of incidence is to rotate the bodies of interest within the fluid domain; however, this presented a problem, as testing revealed that setting the propeller's rotation axis to an axis not parallel with one of the three principal axes of the numerical coordinate system caused numerical instability. Presumably, this was due to slight imperfections in the location of the rotating region due to accumulating floating-point error. This could have caused the interfaces between the rotating and fixed regions to no longer be coincident, inducing errors.

To resolve this problem, the entire fluid domain itself was instead rotated 10° , with the bodies remaining aligned with the numerical coordinate system's principal axes. This resulted in the domain depicted in Fig. 4 and resolved the previous issues with numerical stability.

C. Meshing

Meshing was performed in ANSYS Workbench Meshing. Due to the complex 3D geometry, an unstructured tetrahedron-dominant mesh was used as the basis for spatial discretization. Although this decreased the orthogonal quality of the grid, it allowed a wider variety of discretization techniques to be used (e.g. inflation without multizone meshing), while still maintaining acceptable cell aspect ratios and skewness.

The fixed and rotating regions were meshed as separate parts, as there needed to be a way to associate rotating and fixed cells to distinct cell zones. These two meshes are visible in Fig. 5. These cell zones were associated with each other by adding a "Bonded" contact during the meshing process; this was later referenced during the solution.

Orthogonal quality was maintained in certain areas of interest by seeding the unstructured mesh with structured quadrilateral face meshing. Specifically, this technique was implemented on the upper and lower surfaces of the wing, the upper and lower surfaces of the propeller airfoils, and the interfaces between the rotating and fixed regions. These structured face meshes are visible in Fig. 5. This technique, when combined with local inflation at these faces, ensured that flow near these regions of interests maintained high orthogonal quality.

Fig. 4 The rotated fluid domain of the final geometry

(a) Fixed region mesh

(b) Rotating region mesh

Fig. 5 The two meshes used to construct the sliding mesh model

As mentioned previously, inflation layers were used in order to more accurately capture boundary-layer effects. Inflation was performed on all faces of the wing, nacelle, and propeller.

Due to computational limitations, a turbulence model with wall functions was selected. Therefore, a corresponding

nondimensional wall distance y^+ of 100 was targeted for the wing and nacelle, a value that was outside the viscous sublayer, but definitively within the log-law layer where standard wall functions are valid. On the wing and nacelle, this y^+ corresponded to a first-layer thickness of approximately 1 mm. This inflation is visible in Fig. 6 and near the front of the nacelle in Fig. 5a. On the propeller's faces, a y^+ of 50 was targeted, in an effort to more carefully resolve the vorticity induced by the propeller's motion. This corresponded to a first-layer thickness of approximately 50 μm (much lower due to the propeller's high rotational speed).

The boundary layers were extended 15 layers, which, with a y^+ near 100, almost certainly extended past the boundary layer. However, these large inflation layers were used because they maximized orthogonal quality near the regions of interest, due to the aforementioned structured face meshing.

Fig. 6 The inflation layers used to resolve the airfoil's boundary layer

For the overall unstructured mesh, a curvature-based size function was used in order to accurately capture the geometry of the many curved surfaces in the analysis. A relatively low growth ratio of 1.12 ensured smooth mesh transitions towards the extremities of the domain.

Size constraints were enforced at the leading and trailing edges of lifting surfaces, as well as at the mesh interface. The size function was constrained to 3 mm at the wing's leading and trailing edges and 0.4 mm at the propeller's leading and trailing edges; these fine elements allowed better capture of the vorticity shed by the lifting surface. The cylindrical mesh interface was set to have a 5 mm tangential sizing at the outer edge and a 5 mm radial sizing. The listed settings resulted in a fine mesh of approximately 3.75 million fluid elements.

D. Physics Setup

The problem was set up in ANSYS Fluent, a commercial CFD code that has the capability to simulate a wide variety of flow-related phenomena. The code was based on a pressure-based, transient, implicit Reynolds-Averaged-Navier-Stokes (RANS) solver. The solver used a realizable $k-\epsilon$ turbulence model for closure between the mesh scale and the eddy dissipation scale. This turbulence model was selected for its convergence robustness, and because it provided a good balance between computational cost and simulation accuracy. The $k-\omega$ Shear Stress Transport (SST) and Reynolds Stress models were also considered due to the highly swirling flow, although computational cost precluded their use.

Enabling enthalpy conservation and using an ideal gas model for compressible flow induced numerical instability and residual divergence, so the flow was assumed to be incompressible. This eliminated the need to track enthalpy conservation and resulted in minimal error, as the freestream Mach $M_\infty = 0.10$. Due to adiabatic flow conditions, incompressibility also assumed our flow was isothermal, so a constant viscosity value (taken from air at 25 °C) was used. It is important to add a disclaimer that while the assumption of incompressibility was valid for the wing and nacelle, it may have induced small errors at the propeller tips, which operate at $M_{tip} \approx 0.4$. However, because both the proposed and conventional configurations were simulated using identical parameters, it was predicted that this error would roughly cancel when the differences between scenarios were calculated.

The fixed and rotating cell zones were connected using the previously-established contact between them. The rotating zone was set to spin using the sliding mesh ("mesh motion") model, with the zone rotating in the direction corresponding to the propeller's blades. The rotating zone (containing the propeller) was set to rotate at a speed of 1164.7 rad/sec, corresponding to the propeller advance ratio $J = 0.8$ to match the Dimchev wind tunnel data [9].

Boundary conditions were set as follows:

- The inlet face was held at a fixed velocity of 35 m/s normal to the inlet, corresponding to the freestream velocity of the Dimchev data. Turbulence parameters were made quite laminar in an attempt to mimic wind tunnel conditions, with turbulent intensity of 0.5% and a turbulent viscosity ratio of 2.

- The outlet face was held at a fixed pressure of 101,325 Pa (ambient pressure).
- The centerline wall was set as a symmetry boundary condition, as mentioned previously.
- The three remaining outer walls were set as a free-slip condition, which was essentially a wall with a fixed shear stress of zero. This was numerically identical to the symmetry condition, as zero normal gradients were enforced for all flow variables.
- At all other walls, the no-slip condition (with standard wall function corrections) was enforced.

Reference values for force nondimensionalization were set as follows: density $\rho = 1.225 \text{ kg/m}^3$, area $S = 0.078 \text{ m}^2$ (the projected wing planform area), and velocity $u_\infty = 35 \text{ m/s}$.

E. Solution

ANSYS Fluent was again used to iterate this problem to a steady-state condition. The simulation was conducted in double precision and parallelized across 8 hyper-threaded cores of an Intel i7-6700K processor. Both the proposed and conventional configurations required about 5 days of computation time each on this machine to reach steady-state.

With the pressure-based solver, a fully-coupled scheme was used to relate the pressure and velocity fields. Although this increased computational cost per iteration, it drastically increased the numerical stability and rate of convergence per iteration compared to other schemes (in particular, the PISO algorithm that is more commonly used for unsteady problems). Gradients were calculated using the least squares cell-based method, and pressure interpolation was performed with the PRESTO (PREssure STaggering Option) scheme, which has been validated for the large pressure gradients in highly swirling flows [10].

To decrease the numerical diffusion caused by the lower orthogonal quality of a tetrahedron-dominant mesh, higher order upwind schemes were used. In particular, all convected flow quantities (momentum, turbulent kinetic energy k , and turbulent dissipation rate ϵ) were resolved using a third-order MUSCL (Monotone Upstream-centered Schemes for Conservation Laws) scheme. Time discretization was also higher-order; a bounded-second-order-implicit scheme used as the transient formulation.

The implicit flow Courant number was set to a moderate value of 200. Relaxation factors for momentum and pressure were set at 0.8, and those for k and ϵ were set at 0.75. Convergence was assessed by monitoring conservation residuals as well as the steadiness of nondimensionalized lift and drag on the body.

The cells were given a uniform initialization across the entire domain, taken from the inlet flow values. Fixed time-stepping was used, because the varying-length timesteps of an adaptive model caused discontinuities due to the second-order time discretization method. The fixed timestep of 0.0001 seconds corresponded to approximately 54 timesteps per rotation of the propeller blade. Prior testing indicated that timesteps finer than approximately 40 timesteps per iteration were able to capture the propeller's helical wake, so this timestep length was considered sufficient. 20 iterations were conducted per timestep, which was typically sufficient for all conservation residuals to decrease by at least four orders of magnitude from their initialized values.

The simulations required approximately 0.15 seconds of flow time each in order to reach steady-state in both lift and drag. This corresponded to 1500 timesteps, or approximately 30,000 iterations each. Convergence was achieved, as shown by the plots of engineering quantities of interest in Fig. 7. The discontinuities in the convergence history of the outboard-up case (Figs. 7c and 7d) were the result of gradually stepping up the order of the discretization schemes; it was later determined this was not necessary for numerical stability, so the inboard-up case has no such discontinuities.

F. Results

Table 2 Calculated lift and drag coefficients at $\alpha = 10^\circ$, as a function of propeller direction

Scenario	C_L	C_D
Proposed (inboard-up)	0.8045	-0.1332
Conventional (outboard-up)	0.6814	-0.1267

The results in Table 2 were obtained at steady-state, averaged over 0.01 sec. Note the negative drag coefficient in both scenarios, indicating positive net thrust from the propeller. Based on the data obtained, the proposed configuration offers an **18.1% increase in lift** and a **5.1% increase in net thrust** over the conventional configuration.

The helical vortex from the propeller can be shown to be well-resolved by plotting an isosurface of the Q-criterion, the second invariant of the characteristic equation of the $\nabla \vec{u}$ tensor. As shown in Fig. 8, the vortices remain tight and

(a) Proposed (inboard-up) lift coefficient over time

(b) Proposed (inboard-up) drag coefficient over time

(c) Conventional (outboard-up) lift coefficient over time

(d) Conventional (outboard-up) drag coefficient over time

Fig. 7 Convergence history, inboard-up (top) and outboard-up (bottom)

well-defined until their breakdown over the wing, indicating minimal numerical viscosity.

As seen in Fig. 9, the proposed (inboard-up) flow direction results in a much smaller region of spanwise flow compared to the conventional case, qualitatively confirming the functionality of Active Wingtip Vortex Cancellation.

Another interesting result visible in Fig. 7 is that there is a periodic component to the steady-state solution as well. This periodic component has a fundamental frequency of 741.5 Hz, which is exactly four times the rotational frequency of the propeller ($1164.7 \text{ rad/sec} = 185.4 \text{ Hz}$, corresponding to the specified advance ratio $J = 0.8$). It is hypothesized that these are periodic effects of the helical vortex from the four-bladed prop. This fundamental frequency of 741.5 Hz corresponds to a period of 1.35 ms. The timestep used in this simulation was 0.1 ms, meaning there were approximately 14 samples per period. Thus, in accordance with the discrete-time sampling requirements posited by the Nyquist-Shannon sampling theorem, this periodic unsteady effect has been adequately captured.

G. Validation

The results table was augmented with the Dimchev wind tunnel data (Table 4.4 in [9]), resulting in Table 3. It can be seen that, in both cases, calculated lift values were approximately 20% higher than the corresponding wind tunnel values, and the calculated drag values matched the wind tunnel values very closely.

By far the most likely cause of this lift discrepancy was inaccurate geometry; the Dimchev paper failed to specify several key dimensions of the geometry, such as propeller tip chord and twist, the nacelle-propeller interface, and the nacelle trailing edge shape. It is also unclear whether the wing was modeled after a NACA 642-015 or NACA 642-015A airfoil. Surface roughness was also entirely unknown. In all cases, a best-estimate was made visually.

In addition to this, many unknowns from the wind tunnel testing could have contributed to the error. Inlet turbulence parameters were not specified in the data, so an estimate based on rough correlations from low-subsonic wind tunnels

Fig. 8 Vortex core regions in the proposed (inboard-up) case; Q-criterion isosurface at $6E-5$, colored by absolute helicity

Fig. 9 Spanwise flow velocity (positive-outboard) at the wingtip plane for the proposed (left) and conventional (right) cases

was made. In addition, Dimchev used analytical corrections for solid blockage, wake blockage, slipstream blockage, lift interference, and streamline curvature [9]; this large number of compounding corrections could have contributed a considerable amount of uncertainty to the measurement. The Dimchev paper also failed to explicitly state the reference quantities used, so a discrepancy in nondimensionalization of coefficients is also possible.

However, even with that said, it is important to remember that the key design metric is not so much the actual lift and drag itself, but rather the relative *change* in lift and drag between the proposed and conventional configurations. Using this criterion also mitigates the effect of the previously-listed unknowns, due to similarity between the conventional and proposed cases. The relative increases in lift and drag are listed in the bottom row of Table 3; here, an extremely close match ($\sim 1\%$) between the CFD calculations and the wind tunnel data can be made.

Table 3 Comparison of calculated quantities with wind tunnel data from [9]

Scenario	C_L , calculated	C_D , calculated	C_L , wind tunnel [9]	C_D , wind tunnel [9]
Proposed (inboard-up)	0.8045	-0.1332	0.6668	-0.1337
Conventional (outboard-up)	0.6814	-0.1267	0.5660	-0.1290
Change between proposed and conventional	+18.1%	+5.1%	+17.8%	+3.6%

III. Conclusion

An exotic aircraft configuration, consisting of a low-aspect-ratio conventional small UAV with large wingtip-mounted propellers spinning inboard-up, was proposed. This design, which has only become viable in recent years due to advances in electric motor and battery technology, achieves considerable increases in lift-to-drag ratio by means of active wingtip vortex cancellation with its large propellers.

Various benefits of such a design were outlined, such as increased lift, improved effective span efficiency, decreased induced drag, decreased skin friction drag, extremely wide flight envelopes, high maneuverability, and low observability.

A 3D, transient, viscous, sliding mesh, RANS CFD analysis was performed to validate these claims, in particular, the claims of increased lift and decreased drag from active wingtip vortex cancellation at these scales. The proposed configuration was compared against a baseline ("conventional") configuration with outboard-up propellers, in order to isolate the proposed mechanism of active wingtip vortex cancellation.

The CFD analysis indicated that the proposed case achieved a **18.1% increase in lift** and **5.1% increase in net thrust** when compared to the conventional case, confirming the theoretical predictions. Wind tunnel data from a nearly identical case measured a 17.8% increase in lift and a 3.6% increase in net thrust, so the CFD results matched this metric to within approximately 1%. This extremely close match validates the sliding mesh CFD model for use in analyzing active wingtip vortex cancellation, opening the doors for this to be used as a tool when designing UAVs that take advantage of this fascinating aerodynamic phenomenon.

Acknowledgments

The author would like to thank Dr. Agarwal for his immensely useful guidance during this research. In addition, the author would like to thank the graduate students of Dr. Agarwal's CFD lab, particularly Han Gao, for providing such a welcoming and educational environment to grow as an undergraduate researcher. Finally, the author would like to thank Kevin Hainline, who has been a great mentor during the past few years. Images used courtesy of ANSYS, Inc.

References

- [1] John D. Anderson, Jr., *Fundamentals of Aerodynamics*, 4th ed., McGraw-Hill, New York, 2011.
- [2] Raymer, D. P., *Aircraft Design: A Conceptual Approach*, 2nd ed., American Institute of Aeronautics and Astronautics, Washington, D.C., 1992.
- [3] Smithsonian National Air and Space Museum, "Vought V-173 'Flying Pancake'," , 1998. URL <https://airandspace.si.edu/collection-objects/vought-v-173-flying-pancake>.
- [4] Ginter, S., *Chance Vought V-173 and XF5U-1 Flying Pancakes*, Steve Ginter Publishing, Simi Valley, CA, 1992.
- [5] San Diego Air and Space Museum Online Image Archive, "Vought V-173," , March 2010. URL <https://www.flickr.com/people/sdasmarchives/>, catalog No. 00072646. Navy Photograph. Public Domain.
- [6] Drela, M., "XFOIL," , 2013. URL <http://web.mit.edu/drela/Public/web/xfoil/>.
- [7] Moore, M. D., Goodrich, K., Viken, J., Smith, J., Fredericks, B., Trani, T., Barraclough, J., German, B., and Patterson, M., "High-Speed Mobility through On-Demand Aviation," AIAA 2013-4373, Aviation Technology, Integration, and Operations Conference, Los Angeles, CA, 2013.
- [8] Govindarajan, B. M., "Contributions Toward Understanding the Effects of Rotor and Airframe Configurations On Brownout Dust Clouds," Ph.D. thesis, University of Maryland, 2014.
- [9] Dimchev, M., "Experimental and numerical study on wingtip mounted propellers for low aspect ratio UAV design," Master's thesis, Delft University of Technology, Department of Aerospace Engineering, March 2012.
- [10] ANSYS® *Fluent, Release 18.1.0, Help System, FLUENT Theory Guide*, ANSYS, Inc., 2017.