RANS-Based Aerodynamic Shape Optimization of a Wing Considering Propeller–Wing Interaction

Shamsheer S. Chauhan * and Joaquim R. R. A. Martins†

University of Michigan, Ann Arbor, MI, 48109

The recent growth of interest in the development of hybrid-electric and electric aircraft has led to a renewed focus on the design and optimization of propeller aircraft. Considering propeller–wing interaction for wing-mounted propellers provides the opportunity to design aircraft that take advantage of aerodynamic and propulsion-integration benefits. We carry out RANS-based aerodynamic shape optimization of a wing with an inboard-mounted propeller with the objective of minimizing drag during cruise. We use an actuator-disk approach to model the propeller and use gradient-based optimization. We find that, for the configuration we study, optimizing the wing with the propeller model included in the simulations provides little benefit over optimizing the wing without the propeller model included in the simulations (reduction of 1 drag count).

I. Introduction

The recent growth of interest in the development of hybrid-electric and fully electric aircraft has led to a renewed focus on the design and optimization of propeller aircraft [1–10]. Along with providing benefits related to emissions, procurement cost, and maintenance, electric motors allow distributing propulsion on an aircraft with greater ease compared to combustion engines, which further allows taking advantage of performance benefits from propulsion integration [1]. For example, distributing propellers across a wing can allow sizing smaller wings for conventional and short takeoff and landing (C/STOL) aircraft [1]. As another example, placing propellers at wing tips can result in span efficiency or propulsive efficiency improvements [1, 11].

When a propeller is placed near a wing, the flow induced by it affects the lift of the wing as well as both the profile and induced drag of the wing [11–14]. The details of the flow around the wing and consequently its performance change depending on the location of the propeller and on the direction of rotation of the propeller. When a propeller is placed in front of a wing, the flow seen by the wing sections behind the propeller has higher speeds as well as tangential velocity components (or swirl). These propeller-induced axial and tangential velocity components result in spanwise variations of both the effective flow speeds and angles of attack of the wing sections. Not only does this affect the lift and drag of the sections immediately behind the propeller but also the lift and drag distribution across the entire wing [11, 13, 14]. A propeller can also affect the performance of a wing by influencing the boundary layer. The disturbed flow behind a propeller can make the boundary layer over the wing more turbulent or cause it to cycle between laminar and turbulent transitional states [15], and the static-pressure gradients in the slipstream can also influence boundary layer development. On the other hand, when a propeller is placed behind a wing, the flow seen by the wing sections in front of the propeller have higher speeds (lower than when the propeller is placed in front) but little swirl [14]. Additionally, if the propeller is vertically offset relative to the wing, vertical velocity components related to streamtube contraction can also affect the performance of the wing [12].

At the same time that the flow induced by a propeller affects the performance of a wing, the flow induced by the wing, as well as other changes to the flow field caused by drag or blockage effects, affect the performance of the propeller, primarily by altering the inflow to it [12, 14]. Whether this results in an improvement or loss in performance of the propeller also depends on the location of the propeller [12]. Generally, the impact on the performance of the propeller is greater when it is placed behind a wing.

Considering propeller–wing interaction and accounting for the effects on performance of both the wing and the propeller allow more accurately predicting and maximizing the performance of an aircraft with wing-mounted propellers [1, 7, 14]. There are many approaches of varying complexity and fidelity that have been used to model the interaction between propellers and wings. One approach is to connect a vortex-lattice method (VLM) model to a propeller model that provides induced velocities such as a blade-element momentum (BEM) model [8, 10, 14, 16, 17].
Another approach is modeling a propeller as an actuator disk in a computational fluid dynamics (CFD) simulation (typically Euler or RANS-based CFD) by distributing time-averaged propeller forces over a disk using source terms or boundary conditions [5, 18–20]. This approach allows using steady CFD simulations without modeling the detailed propeller geometry. Another similar approach is to use a rotating actuator line within CFD simulations for unsteady simulations [20]. To capture more detail, a fully unsteady simulation using RANS-based CFD with a three-dimensional rotating propeller geometry is an option [20]. However, this results in very high computational cost.

The approach of using an actuator disk with CFD to model propeller–wing interaction has been shown to provide accurate predictions for the time-averaged performance [19, 20]. When computing propeller forces for the actuator disk, using a blade-element method that uses the inflow to the disk from the CFD simulation allows modeling the mutual interaction between the propeller and the wing and can help improve the accuracy of the predictions [19]. The relatively low cost of an actuator-disk approach compared to a fully unsteady approach makes it a tractable option for RANS-based design studies and optimization.

There are a small number of published studies that optimized wings while accounting for propeller–wing flow interaction. Kroo [13] presented a study to find the optimal lift distributions that minimize the induced losses for a wing with a propeller (in both tractor and pusher configurations). The results show that the optimal lift distribution for a wing with a propeller differs from the elliptical lift distribution that is optimal for an isolated wing. Additionally, significant differences are shown to exist between the optimal lift distributions with inboard-up, outboard-up, and counter-rotating propellers. Kroo [13] also concluded that losses associated with swirl may be recovered with equal increases in efficiency for tractor and pusher configurations. The limitations of the approach used by Kroo [13] include the relatively low order of the models, neglecting viscous effects, and the optimization of the lift distribution instead of geometric design parameters.

Veldhuis and Heyma [21] presented optimization results for a rectangular wing (aspect ratio = 5.3; \(C_L = 0.9\)) with a tractor propeller with the objective of minimizing drag subject to a lift constraint with wing-twist design variables. They used a solver that uses lifting-line theory to model lifting surfaces, and they used the Trefftz-plane method and a viscous-drag estimation method to compute the drag. Their results show negligible differences in drag between minimizing only the induced drag and minimizing the sum of the induced drag and viscous drag (less than 0.1 drag count). They also optimized a propeller-wing configuration based on the Fokker 50 and predicted induced-drag reductions of 13 drag counts with an inboard-up propeller and 8 drag counts with an outboard-up propeller, relative to an optimized wing with no propeller (\(C_L = 0.6\) at Mach 0.35). Their results show increased twist behind the down-going-blade region and decreased twist behind the up-going-blade region (compared to an optimized wing with no propeller). The limitations of their approach include the relatively low order of the models and the limited design freedom (only twist design variables).

More recently, Alba et al. [7] constructed a model for coupled propeller–wing interaction and carried out wing optimization with planform and airfoil-shape design variables. They also included weight and performance models for a multidisciplinary optimization. They modeled the wing (using a quasi-3D approach with a VLM code) and the propeller (using XROTOR) in isolation but used a coupling method to account for the mutual interaction. They concluded that adjusting wing twist and camber to align the wing sections with the propeller swirl is beneficial. For an aircraft based on the Tecnam P2012, their optimization results predict fuel-savings of up to 7% relative to the baseline design. The limitations of their approach include the use of relatively low-order models and the limited number of design variables.

In a recent thesis, Pedreiro [22] presented an aerodynamic shape optimization study for a wing with a tractor propeller (Mach 0.43; \(C_L = 0.48\)), based on the EMB-120 Brasilia, using RANS CFD (CFD++) with an actuator disk and a blade-element code for the propeller loads. Six design variables were used to optimize the twist and thickness-to-chord ratios of the portion of the wing behind the propeller, and a drag reduction of 3 drag counts was obtained. However, Pedreiro [22] used a design-of-experiment approach which, because of the poor scaling of such approaches with the number of design variables, limits the number of design variables that can be used and the ability to carry out a large number of optimization studies.

For this paper, we use an actuator-disk approach and RANS CFD to optimize a wing with an inboard-mounted propeller. We use gradient-based optimization and a modified version of ADflow, which is an open-source CFD solver with an adjoint implementation for efficient gradient computation. This allows considering a large number of design variables for the shape optimization and also allows carrying out multiple optimization studies with manageable computational cost. Note that for this study we only consider the effects of the propeller on the wing and do not account for the effects of the wing on the propeller. This work is a stepping stone towards a coupled CFD model for mutual propeller–wing interaction that can be used for high-fidelity simulations and optimization.
II. Computational Tools

Optimization with RANS CFD requires a multidisciplinary framework that is computationally efficient due to the high computational cost of the simulations. Additionally, since we use a relatively large number of design variables (∼ 200), gradient-based optimization is necessary to make the problem tractable. Therefore, we use the open-source aerodynamic modules of the MDO of Aircraft Configurations with High fidelity (MACH) framework\(^*\) [23].

A. Aerodynamic Solver

We use the open-source RANS CFD code ADflow [24–26] and modify it to use an actuator-disk approach to simulate propeller–wing interaction. The propeller model is described further in Section II.E. ADflow is a second-order finite-volume CFD solver and solves the RANS equations on structured multiblock meshes. It is also capable of handling overset meshes. For this work, we use the Spalart–Allmaras turbulence model and steady simulations.

ADflow uses a discrete adjoint implementation via automatic differentiation to efficiently compute the derivatives of the aerodynamic functions of interest with respect to a large number of design variables. This has enabled several RANS-based optimization studies [27–31].

B. Geometry Parameterization

We use pyGeo, MACH’s geometry manipulation module, to parameterize and manipulate the wing shape using a free-form deformation (FFD) approach [32]. In this approach, the surface of the wing is embedded in a grid of control points, and the changes made to the control points are transferred to the embedded surface using a B-spline mapping. When the control points are moved, the shape inside the volume deforms in a continuous manner, giving the optimizer control over the twist and cross-sectional shapes (as well as the planform shape which in not varied for this paper).

C. Mesh Movement

As the surface geometry is deformed using the FFD control points, IDWarp, MACH’s mesh warping module, is used to deform the original volume mesh during optimization to account for the changes to the surface. IDWarp uses an inverse distance weighting method, which helps preserve the quality of the mesh as the surface is deformed.

D. Optimizer

We use SNOPT [33], a quasi-Newton gradient-based optimizer to solve the optimization problems for this paper. SNOPT handles large-scale nonlinear optimization problems with thousands of constraints and design variables well and is suitable for aerodynamic shape optimization [23, 28–31]. SNOPT is wrapped with pyOptSparse [34] for use with the MACH framework. The major optimality and feasibility tolerances for SNOPT are set to \(1 \times 10^{-5}\) for all the optimization cases in this paper.

E. Propeller Model

We model the propeller using an actuator-disk approach in which forces equal and opposite to time-averaged propeller loads are applied to volume cells in a specified region in the same manner as body forces [18]. To do this, force terms are added to the momentum equations, and corresponding terms are added to the energy equations for the specified cells. In general, the propeller loads can be obtained from various sources such as analytical models [35], blade-element methods [19], or CFD simulations of an isolated propeller [20].

We use the following simple models for the distribution of the axial and tangential loading on the propeller [35]. The radial distribution for the axial force is given by

\[
f_x = F \dot{F}^m \left( \frac{a - \dot{r}}{a} \right)^n
\]

where \(f_x\) is the axial force per unit length, \(\dot{F}\) is a reference value that is adjusted to obtain the required total force, \(\dot{r}\) is defined by Eq. (2), and \(a, m, n\) are parameters that control the shape of the distribution.

\[
\dot{r} = \frac{r - r_{in}}{R - r_{in}}
\]

\(^*\)\url{www.github.com/mdolab/MACH-Aero}
Here, \( r \) is the radial distance from the axis of rotation, \( r_{in} \) is the inner radius of the propeller (usually the radius at which the blades connect to a hub or spinner), and \( R \) is the outer radius of the propeller. The radial distribution for the tangential forces is given by

\[
f_\theta = f_x \left( \frac{P/D}{\pi (r/R)} \right)
\]

(3)

where \( f_\theta \) is the tangential force per unit length, and \( P/D \) is the propeller pitch-to-diameter ratio.

Since Eqs. (1) and (3) provide continuous distributions, we calculate the forces for each cell in the specified region based on their volumes and the distances between their cell-center locations and a specified propeller axis (i.e., \( r \)). The force per unit volume that is used for this is given by

\[
f_x = \frac{f_\theta}{2\pi rt}
\]

(4)

where \( t \) is the total thickness of the disk of volume cells to which the forces are applied. The reference force, \( \widetilde{F} \) (which is not the total force), is set to a value that makes the total axial force applied equal to the desired total force.

III. Validation Cases

A. Geometry and Specifications

To validate the implementation of the propeller model described in Section II.E, we compare simulation results to experimental data provided by Veldhuis [14]. The configuration is a NACA 642-A015 wing with a tractor propeller mounted near the midspan. Figure 1 shows a planform view of the configuration. The wing is unswept, untwisted, and untapered. The propeller axis is located 0.3m from the root of the wing and lines up with the chord lines of the wing sections.

![Fig. 1 Geometry of the experimental configuration (image from Veldhuis [14])](image)

We compare simulations results to 0 and 4 deg angle-of-attack cases tested by Veldhuis [14]. The propeller has an inboard-up rotation for these cases. The Reynolds number for these cases is \( 0.8 \cdot 10^6 \) and the dynamic pressure is 1500 Pa [14]. This translates to a flight speed of 49.5 m/s at sea level conditions (i.e., Mach 0.145). The advance ratio for the propeller is 0.85, and the thrust coefficient is 0.168 [14]. This translates to a thrust of 38.9 N. Veldhuis [14] mentioned that the thrust coefficient is estimated using a bookkeeping approach (measuring forces with and without the propeller on) and using pressure measurements.

B. Simplified Geometry

We use a simplified version of the wing geometry shown in Fig. 1. We do not model the nacelle, and the leading edge of the wingtip is not rounded as shown in Fig. 1. Our simplified wing geometry is shown in Fig. 2a. We obtained
the airfoil coordinates from the online Airfoil Tools† database and fit the points using B-splines. The surface geometry was created using MACH’s pySpline and pyGeo modules.

C. CFD Volume Meshes

We use three mesh refinement levels for the validation simulations to study how the results change as the mesh is refined. We name these meshes the Level 1 (L1), Level 2 (L2), and Level 3 (L3) meshes. The L1 mesh is the finest and the L3 mesh is the coarsest.

For the actuator region, we use a structured multiblock cylindrical volume mesh and apply the forces to a subset of the cells in the middle of this mesh. Figure 2b shows the outlines of the blocks of the cylindrical volume mesh for the L2 refinement. Forces are applied to the cells in this cylindrical mesh that lie inside the disk shown in Fig. 2a. The wing has a separate structured multiblock volume mesh as shown in Fig. 2b (for the L2 refinement). The wing volume meshes were generated by extruding wing surface meshes using MACH’s pyHyp module (Fig. 2a shows the surface mesh for the L2 refinement). The wing surface meshes were generated using ICEM-CFD.

The cylinder and wing volume meshes are overset with a background mesh as shown in Fig. 3a. The background mesh consists of a Cartesian volume mesh in the region where the cylinder and wing meshes are located, and the outer walls of this Cartesian grid are extruded to the farfield by 20 span lengths to give a hemispherical domain (as shown in Fig. 18 in the Appendix). The cylindrical volume mesh is larger than the disk inside which the actuator forces are applied (see Fig. 2) to allow it to be overset with this background mesh.

Figure 1 shows that the rear face of the disk swept by the propeller is located 0.202 m in front of the leading edge of the wing. In our model, the midplane of the cells that are selected for the actuator zone is located at this distance in front of the wing. For the L2 and L3 meshes, the forces are applied to a circular region of cells that are one layer thick, and for the finer L1 mesh, the forces are applied to a circular region of cells that are two layers thick. The radius of the disk that selects the cells for the actuator zone is the propeller radius of 0.118 m, and the thickness of the actuator zone is 9.1 mm.

Table 1 lists the numbers of cells in the three overset meshes. The wing surface mesh for the L2 refinement was generated by coarsening the L1 mesh by removing every other node, and the wing surface mesh of the L3 refinement was generated by coarsening the L2 mesh by again removing every other node. These surface meshes and the grid spacings of the Cartesian volume mesh that surrounds the wing and cylindrical volume meshes are shown in Fig. 17a in the Appendix. To generate the wing volume meshes, the wing surface meshes were extruded to 0.1 m around the wing using pyHyp. The L1, L2, and L3 wing volume meshes have 80, 40, and 30 layers of cells extruded around the wing.

---

respectively. The off-wall cell thicknesses for the L1, L2, and L3 meshes are set to flat-plate \( y^+ \) values of 0.5, 1, and 2, respectively (for the flow conditions of the optimization cases these would be approximately double). Figure 17b in the Appendix shows the L1, L2, and L3 cylindrical volume meshes.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Total number of cells</th>
<th>Total number of computation cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>L1</td>
<td>10,621,440</td>
<td>10,085,277</td>
</tr>
<tr>
<td>L2</td>
<td>1,343,520</td>
<td>1,263,010</td>
</tr>
<tr>
<td>L3</td>
<td>509,152</td>
<td>476,049</td>
</tr>
</tbody>
</table>

### D. Propeller Model Inputs

For the adjustable parameters mentioned in Section II.E, we use \( a = 1, m = 1, \) and \( n = 0.2 \) which gives a loading distribution that is highest near the tip. For the propeller we are modeling, \( P/D \) ranges from 0.9 to 1.1 for the \( r/R = 0.45 \) to \( r/R = 0.95 \) portion of the blades [36]. However, we use \( P/D = 0.85 \) for a better match with the experimental data. CFD simulations by Stokkermans et al. [20] for this propeller show that the thick root sections of the blades from \( r/R = 0.35 \) to the root generate negative thrust due to flow separation. Therefore, for our model, we set \( r_{in} = 0.35R \) and use the axial force distribution from Eq. (1) between \( r = R \) and \( r = 0.15R \) (the roots of the propeller blades are located at \( 0.15R \)) which gives negative thrust between \( r = 0.35R \) and \( r = 0.15R \). We set the axial forces to zero for \( r < 0.15R \). For the region between \( r = 0.35R \) and \( r = 0.15R \), we reduce the axial forces given by Eq. (1) by multiplying them by a factor of 0.25, as the results of Stokkermans et al. [20] show a change in slope after the axial force becomes 0 at approximately \( r = 0.35R \). This portion of the propeller not generating positive thrust also roughly coincides with the diameter of the nacelle (\( r = 0.3R \)). For the tangential forces, we use the distribution from Eq. (3) only between \( r = R \) and \( r = 0.35R \), and we set the tangential forces to zero for \( r < 0.35R \). The reference force, \( \tilde{F} \), is set to a value that makes the total axial force applied to the region outside \( r_{in} \) equal to the specified total force. We use the region outside \( r_{in} \) because Veldhuis [14] mentioned that the thrust coefficients for the test cases were computed using a bookkeeping approach with pressure measurements, and we consider these thrust coefficients to more likely represent the thrust.

![Fig. 3 Overset meshes](image)
generated by the propeller region outside the nacelle radius of \( r = 0.3R \). Fig. 4 shows the shapes of the resulting distributions using Eqs. (1) and (3).

Fig. 4  Radial distribution of propeller axial and tangential forces for the validation cases

E. Validation Results

Figure 5 compares sectional lift coefficients from ADflow to experimental data for the 0 deg and 4 deg validation cases. The results with the L1 and L2 meshes match the experimental data well for the 0 deg case and reasonably well with some discrepancies for the 4 deg case. The L3 mesh is not able to adequately resolve the spanwise variations.

Fig. 5  Comparison of ADflow results to experimental data from Veldhuis [14]
One discrepancy is the higher lift predicted inboard of the propeller and behind the up-going-blade region for the 4 deg case. One possible physical explanation is that our axisymmetric propeller model does not capture the non-axisymmetric propeller loading that occurs in reality due to the angle of attack. An up-going blade will experience smaller effective angles of attack and lower effective speeds than a down-going blade due to the component of the freestream velocity parallel to the plane of the propeller disk. This results in the up-going blade applying lower axial and tangential forces to the fluid and the down-going blade applying higher axial and tangential forces compared to a case with zero angle of attack. This would explain why our model results in a greater lift at the root, a higher peak inboard, and a slightly shorter trough outboard compared to the experimental data for only the 4 deg case.

Figure 6 shows pressure coefficient ($C_p$) contours on the wing for the 4 deg angle-of-attack case and also shows the shapes of the pressure distributions at four distinct locations on the wing. Behind the up-going-blade region, the $C_p$ curves show a larger suction peak compared to the sections near the root and the tip of the wing. On the other hand, behind the down-going-blade region, there is no suction peak. $C_p$ distributions for two airfoil sections behind the propeller are also compared to experimental data in Fig. 7 for the 4 deg angle-of-attack case. The shapes match, but there is an offset.

Figures 19 and 20 in the Appendix show the $C_L$ and $C_D$ values with the three mesh refinement levels for the 0 deg and 4 deg validation cases. The CFD solver settings and angle of attack are kept the same for the different meshes. These results show that unlike the drag coefficient the lift coefficient does not decrease monotonically as the mesh is refined.

For the optimization studies described in the following sections, we use the L2 mesh, as it provides reasonably good resolution with a significantly lower computational cost than the L1 mesh.
IV. Optimization Problem Descriptions

A. Geometry and Parameterization

For the optimization cases, we use the same wing and propeller configuration used for validation in Section III. We use the L2 mesh described in Section III.C and the propeller-model inputs discussed in Section III.D. We only optimize the wing, and the optimizations begin with the wing untwisted and both the wing and propeller at an angle of attack of 0 deg.

Figure 8 shows the grid of FFD control points that we use to deform and optimize the geometry. The positions of these control points are used as the design variables. The grid has 13 chordwise sections along the span and 8 spanwise sections along the chord. There are 104 FFD control points above the wing and 104 below it (208 in total). The control points at each chordwise section can be rigidly rotated together to twist the wing, and all the control points can be moved individually in the vertical direction (y-direction) to change the shapes of the airfoil sections along the wing. Note that the planform area and planform shape of the wing are not being optimized in any of the cases presented in this paper. We bunched control points closer together over the portions of the wing behind the up-going and down-going-blade regions of the propeller disk for greater control of those portions.

B. Flight Conditions

The flight conditions that we use for the optimization cases are selected to be representative of a reasonable cruise condition. We use a nominal cruise $C_L$ of 0.6 at a Mach number of 0.3 and an altitude of 1500 ft. We also assume that the hypothetical aircraft that the wing belongs to has a cruise lift-to-drag ratio of 10. Based on these numbers, we also specify a total thrust of 55.0 N for the propeller actuator disk. With these cruise specifications, this hypothetical twin-propeller aircraft would be a UAV with a total mass of $\sim$ 110 kg.

C. Primary Optimization Problem Formulations

We carry out both single-flight-point and multiple-flight-point (referred to as multipoint) optimizations. We have two primary single-point optimization cases and one multipoint optimization case.

Table 2 summarizes the optimization problem formulation for the first optimization case, Case wP+T. In Case wP+T, in the presence of the propeller (wP), we optimize only the wing twist (T). The twist is controlled by rigidly rotating the FFD control points of each of the 13 chordwise FFD sections located along the span. There are 16 FFD control points for each chordwise section but only one twist variable each to rotate the sections (about the quarter-chord point of the...
FFD sections). We optimize the twist using all 13 chordwise FFD sections, including the root section, which means that the twist design variables can rotate the wing and effectively change its angle of attack without changing the angle of attack of the propeller disk (as opposed to changing the angle of attack of the complete configuration by changing the freestream flow direction). The optimization objective is to minimize drag subject to a lift constraint of $C_L = 0.6$. Note that the planform area and planform shape of the wing are not being optimized in any of the cases presented in this paper.

Table 2  Case wP+T optimization problem formulation (twist only)

<table>
<thead>
<tr>
<th>Function/variable</th>
<th>Description</th>
<th>Quantity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimize $C_D$</td>
<td>Drag coefficient</td>
<td>13</td>
</tr>
<tr>
<td>with respect to $-10.0 \leq \tau \leq 10.0$</td>
<td>Twist of each FFD section [deg]</td>
<td>13</td>
</tr>
<tr>
<td>subject to $C_L = 0.6$</td>
<td>Lift constraint</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Total design variables</td>
<td>13</td>
</tr>
<tr>
<td></td>
<td>Total constraint functions</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 3 summarizes the optimization problem formulation for the second optimization case, Case wP+T+S. In Case wP+T+S, in the presence of the propeller (wP), we optimize the wing twist (T) and the airfoil shapes (S) along the wing using the FFD control points. The twist is controlled as described earlier, and the airfoil shapes along the wing are controlled by vertically displacing the FFD control points. We limit these additional vertical displacements to $\pm 0.0144$ m (i.e., $\pm 40\%$ of the original airfoil thickness). The optimization objective is to minimize drag subject to a lift constraint of $C_L = 0.6$, which is the same as that of Case wP+T. We also use thickness constraints to prevent the thickness of the wing from decreasing. We use locations on a 10 by 10 grid of points over the wing to enforce these constraints on. These constraints are used because considerations related to the wing structure or packaging of the internal components are not considered here. Additionally, we use constraints on the leading-edge and trailing-edge FFD control points to prevent them from providing redundant control over the twist.

Table 4 summarizes the optimization problem formulation for the third optimization case, Case wP+T+S+mpt, which is a multipoint case. Case wP+T+S+mpt has the same formulation as Case wP+T+S except that we consider three cruise lift coefficients, 0.6 and $0.6 \pm 5\%$, instead of just one. The intention is to check whether the optimization results change significantly when the wing is optimized for a range of lift coefficients around the nominal lift coefficient instead of just at that particular lift coefficient. The optimization objective is to minimize the sum of the drag coefficients corresponding to the three different lift coefficients. Each of the two off-design points ($C_L = 0.6 \pm 5\%$) has an additional angle-of-attack design variable that is used to achieve the required lift with the same wing as the other points. These
Table 3  Case wP+T+S optimization problem formulation (twist and shape)

<table>
<thead>
<tr>
<th>Function/variable</th>
<th>Description</th>
<th>Quantity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimize</td>
<td>$C_D$</td>
<td></td>
</tr>
<tr>
<td>with respect to</td>
<td>$-10.0 \leq \tau \leq 10.0$</td>
<td>13</td>
</tr>
<tr>
<td></td>
<td>$-0.0144 \leq \Delta y \leq 0.0144$</td>
<td>208</td>
</tr>
<tr>
<td>subject to</td>
<td>$C_L = 0.6$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$t/t_{\text{initial}} \geq 1.0$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\Delta y_{\text{LE,upper}} = -\Delta y_{\text{LE,lower}}$</td>
<td>13</td>
</tr>
<tr>
<td></td>
<td>$\Delta y_{\text{TE,upper}} = -\Delta y_{\text{TE,lower}}$</td>
<td>13</td>
</tr>
<tr>
<td>Total design variables</td>
<td></td>
<td>221</td>
</tr>
<tr>
<td>Total constraint functions</td>
<td></td>
<td>127</td>
</tr>
</tbody>
</table>

angle-of-attack variables change the angle of the freestream velocity instead of rotating the wing. We used this approach because of its simpler implementation with ADflow. To match the lift coefficients for the off-design points, these angles of attack will be small (approximately ±0.5 deg) and the resulting incidence angles of the propeller to the freestream are not a major concern. Also note that we did not change the thrust for the three flight points. For a more consistent multipoint optimization, the thrust should be adjusted to match the changes in drag with the different lift coefficients and the new optimized design. However, we did not consider the extra complexity required for this to be necessary for our purposes here.

Table 4  Case wP+T+S+mpt optimization problem formulation (multipoint with twist and shape)

<table>
<thead>
<tr>
<th>Function/variable</th>
<th>Description</th>
<th>Quantity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimize</td>
<td>$C_{D_{p1}} + C_{D_{p2}} + C_{D_{p3}}$</td>
<td></td>
</tr>
<tr>
<td>with respect to</td>
<td>$-10.0 \leq \tau \leq 10.0$</td>
<td>13</td>
</tr>
<tr>
<td></td>
<td>$-0.0144 \leq \Delta y \leq 0.0144$</td>
<td>208</td>
</tr>
<tr>
<td></td>
<td>$\alpha_{p1}$</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>$\alpha_{p3}$</td>
<td>1</td>
</tr>
<tr>
<td>subject to</td>
<td>$C_{L_{p1}} = 0.57$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$C_{L_{p2}} = 0.6$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$C_{L_{p3}} = 0.63$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$t/t_{\text{initial}} \geq 1.0$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\Delta y_{\text{LE,upper}} = -\Delta y_{\text{LE,lower}}$</td>
<td>13</td>
</tr>
<tr>
<td></td>
<td>$\Delta y_{\text{TE,upper}} = -\Delta y_{\text{TE,lower}}$</td>
<td>13</td>
</tr>
<tr>
<td>Total design variables</td>
<td></td>
<td>223</td>
</tr>
<tr>
<td>Total constraint functions</td>
<td></td>
<td>129</td>
</tr>
</tbody>
</table>

D. Baseline Cases for Comparison

To quantify the benefit of optimizing the wing with propeller effects, we also optimize the wing without propeller effects (i.e., without applying forces to the actuator zone) to obtain baseline cases for comparison. The FFD grid used for these baselines cases is the grid used for the wP cases (shown in Fig. 8) with the fifth and eighth chordwise FFD sections, counting from the root, removed (these FFD control points were added to increase the FFD grid density for
greater control over the region behind the propeller for the wP cases). The first baseline optimization case, Case T, has the same formulation as Case wP+T, and the only differences are the different number of FFD control points and that no forces are applied to the actuator zone. The second baseline optimization case, Case T+S, has the same formulation as Case wP+T+S, and again the only differences are the different number of FFD control points and that no forces are applied to the actuator zone. The resulting wings from these baseline optimization cases are then re-simulated with the propeller forces applied to the actuator zone and the wing rotated (using the FFD grid) to achieve the specified $C_L$ of 0.6. This gives us one-to-one comparisons that allow us to quantify the benefit that optimizing a wing with propeller effects considered gives over optimizing the same wing without considering propeller effects.

V. Optimization Results

Figure 9 plots the drag coefficients of the optimized wings ($C_L = 0.6$ for all). As described previously, the baseline cases which were optimized without the propeller (Cases T and T+S) are re-simulated with the propeller and matched to the required lift coefficient to compute these drag coefficient values. These results show that the benefit obtained by optimizing the wing in the presence of the propeller is not significant (1 drag count when using shape design variables). Also, there is a negligible difference between the drag coefficients of the single-point wP+T+S case and the multipoint wP+T+S+mpt case.

![Diagram of drag coefficients](image)

**Fig. 9 Optimized drag coefficients ($C_L = 0.60$ for all)**

Figure 10 shows the optimized twist, thickness-to-chord ratio, and lift distributions along the span of the wing for the different cases. When optimizing with the propeller, the optimizer reduces the wing twist behind the propeller (behind both the up-going and down-going-blade regions). The twist of most of the wing is lower for the cases with shape design variables compared to the cases without them because of cambering. At the wing tips, we obtain large twisting as the optimizer makes the wing tip highly nonplanar and creates a shape resembling a blended winglet. Additionally, for the cases optimized with the propeller, the optimizer takes advantage of weaknesses in the thickness constraints to slightly reduce the thickness of the wing behind the propeller (inboard of the up-going-blade region and directly behind the down-going-blade region). The optimized lift distributions are similar for all the cases and, as expected [13], are not completely elliptical.

Figures 11 and 12 show front and rear views of the optimized geometries. The small differences in geometry due to the propeller effects are visible when comparing the T+S and wP+T+S optimized geometries. Both the T+S and wP+T+S optimized wings have the highly non-planar wingtips, and therefore we can safely conclude that this is not related to propeller effects. Figure 13 shows pressure contours on the optimized wings, and Fig. 14 shows optimized airfoil shapes and their corresponding $C_p$ curves at four locations on the wing. From these two figures we see that with shape design variables the pressure peaks are reduced and the airfoils are cambered.
Fig. 10  Optimized twist, thickness, and lift distributions

Fig. 11  Front view of the optimized wings
Fig. 12  Rear view of the optimized wings

Fig. 13  Optimized wings with $C_p$ contours
VI. Additional Optimization Cases

For reference, we include results for two additional optimization cases which have greater freedom for the shape design variables. These cases, named Cases T+xS and wP+T+xS, do not have thickness constraints, and their shape design variables have larger bounds of ±0.0288 m (80% of the original airfoil thickness) instead of ±0.0144 m for Cases T+S and wP+T+S. Apart from these differences, Case T+xS has the same optimization problem formulation as Case T+S, and Case wP+T+xS has the same formulation as Case wP+T+S. The intention is to verify that optimizing the wing with the presence of the propeller provides little advantage over optimizing it without the propeller and that the small drag reduction that we saw in Section V is not an artifact of overly restrictive thickness constraints or insufficient design freedom for the shape design variables. Note that these additional cases are not practical design cases because of the lack of thickness constraints in the absence of structural and packaging considerations.

Figure 15 plots the drag coefficients for these additional cases (\(C_L = 0.6\) for all) and shows that again there is little difference in drag when optimizing with or without the propeller. Figure 16 shows the optimized twist, thickness-to-chord ratio, and lift distributions for these additional cases compared to the distributions for the wP+T+S case. The optimizer takes advantage of the additional design freedom and reduces the wing thickness to reduce drag. Again, however, the optimized distributions do not differ significantly when optimized with or without the propeller.
Fig. 15  Optimized drag coefficients of the additional cases (Cases T+S and wP+T+S included for comparison)

Fig. 16  Optimized twist, thickness-to-chord ratio, and lift distributions of the additional cases (Case wP+T+S included for comparison)
VII. Conclusion

For this paper, we implemented an actuator-disk propeller model in the CFD solver ADflow, and we carried out RANS-based aerodynamic shape optimization studies for a wing with an inboard-mounted tractor propeller using the open-source aerodynamic modules of the MACH\textsuperscript{§} framework. We presented the results of seven optimization cases which were solved using gradient-based optimization enabled by efficient gradient computation. With the objective of minimizing drag during cruise, the optimizer decreased the wing twist and thickness behind the propeller. However, for the configuration that we studied (the wing of a hypothetical twin-propeller 110 kg UAV), we found that the benefit of optimizing the wing in the presence of the propeller is practically negligible compared to optimizing the wing without it (reduction of 1 drag count). Recommendations for future work include studying how general these conclusions are when considering aircraft of different sizes and configurations.

Appendix: Additional Figures for the Validation and Mesh-Refinement Studies

Figure 17 shows grid spacings for the three mesh refinement levels, and Fig. 18 shows the shape of the complete hemispherical CFD domain of the meshes used in this paper (these are further described in Section III.C). Figures 19 and 20 plot lift and drag coefficients from the mesh refinement study for the validation cases (described in Section III.E).

![Fig. 17 Comparing L1, L2, and L3 grids](image)

**Acknowledgments**

The authors thank Charles Mader, Nicolas Bons, Anil Yildirim, Yingqian Liao, and Eirikur Jonsson for their helpful suggestions and invaluable support with the various computational tools used for this work.

\textsuperscript{§}www.github.com/mdolab/MACH-Aero
Fig. 18  Complete hemispherical domain (L2 mesh)

Fig. 19  Mesh refinement plots for the 0 deg angle-of-attack validation case (N is the number of cells)

Fig. 20  Mesh refinement plots for the 4 deg angle-of-attack validation case (N is the number of cells)
References


