A HIGHER-ORDER FREE-WAKE METHOD FOR AERODYNAMIC PERFORMANCE PREDICTION OF PROPELLER-WING SYSTEMS

A Dissertation in
Aerospace Engineering

by

Julia Ann Cole

© 2016 Julia Ann Cole

Submitted in Partial Fulfillment of the Requirements for the Degree of

Doctor of Philosophy

December 2016
The dissertation of Julia Ann Cole was reviewed and approved* by the following:

Mark D. Maughmer  
Professor of Aerospace Engineering  
Dissertation Advisor, Chair of Committee

Edward C. Smith  
Professor of Aerospace Engineering

Sven Schmitz  
Associate Professor of Aerospace Engineering

Gül Kremer  
Professor of Industrial and Manufacturing Systems Engineering, Iowa State University

Götz Bramesfeld  
Assistant Professor of Aerospace Engineering, Ryerson University  
Special Member

Philip J. Morris  
Professor of Aerospace Engineering  
Head of the Department of Aerospace Engineering

*Signatures are on file in the Graduate School.
Abstract

A new higher-order free-wake (HOFW) method has been developed to enable conceptual design space explorations of propeller-wing systems. The method uses higher order vorticity elements to represent the wings and propeller blades as lifting surfaces. The higher order elements allow for better force resolution and more intrinsically computationally stable wakes than a comparable vortex-lattice method, while retaining the relative ease of geometric representation inherent to such methods. The propeller and wing surfaces and wakes are modeled within the same flow field, thus accounting for mutual interaction without the need for empirical models.

The method was shown to be accurate through comparisons with other methods and experimental data. To ensure the method is capable of capturing an unsteady lift response, it was compared with a Küssner function approximation of the change in two-dimensional lift due to a sharp-edged gust. This study showed excellent agreement with an average error in the HOFW lift response of less than 3% from 0 to 10 semi-chords, but required high time and space resolution. The time-accurate lift response of a propeller-wing system as predicted with the HOFW method was then compared with with fully unsteady CFD. These results showed that the HOFW method can identify the peak frequency and general amplitude of the lift oscillations at high resolution. Due to the high resolution requirements, this mode of analysis is not recommended for use in design studies.

Time-averaged results found using the HOFW method were compared with experimental propeller, proprotor, and propeller-wing system data, along with two semi-empirical methods. The method matched experimental propeller efficiencies to within 4% for lightly loaded conditions. Increases in lift coefficient due to interaction with a propeller for a series of wings as analyzed with the HOFW
method matched the average of those predicted with two semi-empirical methods with an average of 6.5% error for a lightly loaded propeller case. A comparison of HOFW predictions of lift for a more non-conventional propeller-wing system with experimental results over a range of angles of attack showed an average difference of 0.04 in lift coefficient. For this system, predictions in thrust and torque also matched experimental results within 5% over a small angle of attack range (±5°). The method was less successful at predicting the magnitude of drag in comparison with experimental results, but was capable of qualitatively matching trends in drag, both with changes in angle of attack and for variations in design.

Finally, two design studies were conducted to show the practical utility of the method: an investigation of the twist distribution on a large civil tiltrotor wing and an investigation into propeller rotation direction and vertical location on a distributed electric propulsion vehicle. The studies showed that the method is capable, fast, accurate, and robust for performance prediction of propeller-wing systems, and thus appropriate for use in design-space exploration.
# Table of Contents

List of Figures ............................................................ ix
List of Symbols .......................................................... xvii
Acknowledgments .......................................................... xx

Chapter 1
**Introduction and Background** ........................................ 1

1.1 Motivation ............................................................. 1
1.2 Structure of Thesis ................................................... 3
1.3 Background .......................................................... 3
1.3.1 Influence of the Propeller on the Wing ....................... 4
1.3.1.1 The Wake of the Propeller .................................. 4
1.3.1.2 Impact of the Propeller Wake on the Wing ............. 15
1.3.2 Influence of the Wing on the Propeller ....................... 25
1.3.3 Influence of Mutual Interaction on Aircraft Performance .... 30
1.3.4 Summary .......................................................... 34
1.4 State of the Art ....................................................... 35
1.5 Research Objectives .................................................. 41
1.6 Overview of Proposed Method ...................................... 42

Chapter 2
**Theoretical Derivation and Numerical Solution Method** ........ 45

2.1 Overview ............................................................. 45
2.2 Historical Context ................................................... 46
2.3 Theoretical Formulation of Method ................................ 49
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.3.1 Derivation of Governing Equations for Unsteady Potential Flow</td>
<td>50</td>
</tr>
<tr>
<td>2.3.2 The Distributed Vorticity Element</td>
<td>56</td>
</tr>
<tr>
<td>2.4 Modeling of the Lifting Surfaces and Wakes</td>
<td>59</td>
</tr>
<tr>
<td>2.4.1 Lifting Surfaces</td>
<td>59</td>
</tr>
<tr>
<td>2.4.2 Rotating Lifting Surfaces</td>
<td>62</td>
</tr>
<tr>
<td>2.4.3 Compressibility Corrections</td>
<td>62</td>
</tr>
<tr>
<td>2.4.4 Wake Development</td>
<td>64</td>
</tr>
<tr>
<td>2.4.5 Special Kinematic Considerations for Rotating Surfaces</td>
<td>69</td>
</tr>
<tr>
<td>2.5 Numerical Solution Method</td>
<td>71</td>
</tr>
<tr>
<td>2.6 Clarification of Modes of Analysis for Propeller-Wing Systems</td>
<td>81</td>
</tr>
<tr>
<td>2.7 Framework for Rapid Design-Space Exploration</td>
<td>83</td>
</tr>
<tr>
<td>2.8 Summary</td>
<td>84</td>
</tr>
</tbody>
</table>

Chapter 3

Calculation of Forces and Limitations of Method 86

3.1 Overview 86

3.2 Lift 86

3.2.1 Discussion of Unsteady Lift 86

3.2.2 Implementation 91

3.3 Vortex-Induced Drag 95

3.3.1 Trefftz Plane Analysis 97

3.3.1.1 Flowfield Unsteadiness 99

3.3.1.2 Discussion of Analysis 99

3.3.2 Relaxed Wakes vs. Fixed, Drag-Free Wakes 101

3.3.3 Summary of Support of Approach to Induced-Drag Calculation 104

3.3.4 Implementation 105

3.4 Consideration of the Leading-Edge Suction Force 105

3.5 Profile Drag Estimation 107

3.6 Stall Model 110

3.7 Propeller Performance 111

3.8 Thrust and Drag Bookkeeping 114

3.9 Limitations of the Model 116

3.10 Summary 124

Chapter 4

Grid Resolution Studies 126

4.1 Overview 126

4.2 Grid Resolution Properties of Vortex-Lattice Methods 127
## List of Figures

1.1 Definition of axial, tangential, and radial induced velocities due to a propeller. .................................................. 4
1.2 Variation of induced axial velocity as a function of disk loading as defined by Eq. 1.1 at sea level density for varying freestream velocities. The two aircraft examples are provided for context and are calculated at their respective cruise conditions. .................. 6
1.3 Variation of induced tangential velocity as a function of disk loading for a propeller at sea level density for varying freestream velocities as calculated using QPROP [1]. For context, the operating point for a GA aircraft in a sea-level cruise is included. ....................... 8
1.4 Illustration of a vortex-wake model of a propeller neglecting slipstream contraction. .................................................... 9
1.5 Planforms of the two example propeller blades analyzed in Fig. 1.6. 10
1.6 A comparison of the induced velocities for two propellers of differing designs at the same thrust and advance ratio within the propeller plane. ............................................................... 11
1.7 An example of the variation of induced axial and tangential velocities with advance ratio within the propeller plane as calculated with QPROP [1]. ................................................................. 12
1.8 Example of axial induced velocity variation at a single radial location [2] (reprinted with permission of the ASME). ................. 13
1.9 Example of tangential induced velocity variation with axial location [2]. ........................................................................... 13
1.10 Example of induced velocity variations (left - axial, center - tangential, right - radial) with radial location [2]. ..................... 14
1.11 Illustration of the impact of the propeller wake on the wing lift distribution as it depends on propeller rotation direction. Modified and reprinted with permission from Ref. [3]. ....................... 16
1.12 Wind tunnel test results showing the spanwise lift distribution of a straight, untwisted wing (AR = 5.33, α = 4°) with and without the influence of a propeller (J=0.85, C_T = 0.168) [3].

1.13 Change in lift coefficient with propeller streamwise location for a high-speed case (J=1.63, C_T = 0.046) and a low-speed case (J=1.00, C_T = 0.251) as calculated using a modified vortex-lattice method [4].

1.14 Illustration of the interaction between the wing-tip vortex and the wake of a propeller located at the wing tip. The propeller rotating up inboard produces tangential velocities that are opposite of those produced by the wing-tip vortex.

1.15 Experimentally determined wing lift coefficient (upper) and drag coefficient (lower) as a function of propeller vertical location at a low and high thrust condition (α = 4°) [3].

1.16 Local circulation as a function of spanwise location for propeller off and down-inboard rotation (C_T = 0.155).

1.17 Surface pressure coefficients as a function of normalized time at a single chordwise location (x/c=0.04, J=0.37, wing angle of attack = 10°) [5] (reprinted with permission of AIAA).

1.18 Vorticity within the symmetry plane for an 8-bladed propeller-wing interaction captured using PIV [6].

1.19 Comparison of lift and inviscid drag distributions on a wing within a propeller wake for time-averaged and unsteady Euler analysis [7] (reprinted with the permission of AIAA).

1.20 Induced upwash of an NACA 0012 airfoil (V_∞ = 200 ft/s) at α = 5° (upper) and α = 10° (lower) as calculated using Solidworks® Flow Simulation.

1.21 Influence of wing upwash on propeller efficiency as calculated using a blade element method [3]. In this case, the propeller is located one radius in front of the leading edge of the wing (x_P = 1.0R).

1.22 Influence of mutual interaction on power available and power required of an example aircraft.

1.23 Example drag build-up for a conventional turboprop aircraft estimated using empirical methods.

1.24 Example power build-up for a conventional turboprop aircraft estimated using empirical methods.

2.1 A Distributed Vorticity Element (DVE) is a combination of two coplanar, overlapping, second-order lifting lines of opposite orientation [8].

2.2 Simple lifting line whose potential function is given by Eq. 2.15.
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.3</td>
<td>The body-fixed coordinate system.</td>
<td>60</td>
</tr>
<tr>
<td>2.4</td>
<td>An example of the conversion of a simple planar wing geometry to one chordwise row of surface DVEs.</td>
<td>61</td>
</tr>
<tr>
<td>2.5</td>
<td>The propeller pitch angle ($\beta$) is related to the incidence angle ($\epsilon$) via a 90° rotation, as shown for a simple rectangular propeller blade rotating in the positive direction about the x-axis.</td>
<td>63</td>
</tr>
<tr>
<td>2.6</td>
<td>A comparison of unsteady versus steady wake development as shown notionally with a fixed-wake.</td>
<td>65</td>
</tr>
<tr>
<td>2.7</td>
<td>Kelvin’s theorem of vorticity as applied to a steady flow within the context of an aircraft wing.</td>
<td>67</td>
</tr>
<tr>
<td>2.8</td>
<td>Kelvin’s theorem of vorticity as applied to an unsteady flow in the context of an aircraft wing.</td>
<td>68</td>
</tr>
<tr>
<td>2.9</td>
<td>The velocity designated for each SDVE on a propeller blade must be defined such that the SDVE arrives at the correct location in space. This requires a delineation between the rotational velocity used for aerodynamic analysis ($V_{i,r}$) and that used to define the motion of the blade ($V_{i,d}$).</td>
<td>71</td>
</tr>
<tr>
<td>2.10</td>
<td>A flow chart of the HOFW method as modified for analysis of propeller-wing systems.</td>
<td>73</td>
</tr>
<tr>
<td>2.11</td>
<td>An example of the discretization of a simple planar wing geometry using one chordwise row of surface DVEs.</td>
<td>76</td>
</tr>
<tr>
<td>2.12</td>
<td>Flowchart detailing the possible modes of analysis within the HOFW method. Overall default options are indicated in green. The default options for unsteady analysis are outlined in purple.</td>
<td>82</td>
</tr>
<tr>
<td>3.1</td>
<td>Angle and coordinate system definitions for pressure integration on a two-dimensional flat plate.</td>
<td>88</td>
</tr>
<tr>
<td>3.2</td>
<td>Chordwise distribution of circulation at a given spanwise location as it relates to vorticity on the lifting surface represented with five rows of DVEs.</td>
<td>92</td>
</tr>
<tr>
<td>3.3</td>
<td>Flow chart outlining justification for use of the Kutta-Joukowski law in calculation of induced drag for propeller-wing systems with relaxed wakes.</td>
<td>98</td>
</tr>
<tr>
<td>3.4</td>
<td>Definition of velocities and forces for the calculation of profile drag.</td>
<td>107</td>
</tr>
<tr>
<td>3.5</td>
<td>Definition of velocities and forces on a section of a rotating lifting surface.</td>
<td>112</td>
</tr>
<tr>
<td>3.6</td>
<td>Definition of propeller normal force direction.</td>
<td>113</td>
</tr>
<tr>
<td>3.7</td>
<td>The propeller normal force as a percent of the total integrated force in the lift direction for a simple propeller-wing system.</td>
<td>114</td>
</tr>
</tbody>
</table>
3.8 Image of the intersection of the wake and the wing as exists within the HOFW method. The wing and propeller lifting SDVEs are shown in red and the wakes are shown in blue. One location of intersection is highlighted with a gold oval.

3.9 Path of a propeller tip vortex as it traverses a wing according to Johnston and Sullivan [5] (reprinted with permission of AIAA).

3.10 Sketch of the model that would result from splitting the wake around the wing in the HOFW method.

3.11 Stream-lines (shown in blue) over the wing (shown in red) and wake (shown in yellow) calculated at a single time-step of a single wing case with a.) one row, b.) two rows, c.) three rows, and d.) five rows of surface DVEs.

3.12 Stream-lines (shown in blue) over the wing (shown in red) calculated at a single time-step of propeller-wing interaction with a.) one row, b.) two rows, and c.) three rows of surface DVEs. The wake DVEs (for both the wing and propeller) are shown in yellow.

4.1 Geometry of representative propeller-wing system used for convergence studies.

4.2 Wall clock time as a function of increasing spatial resolution for the representative propeller-wing system as calculated using the Penn State Lion-XV computing cluster (Intel Xeon E5-2600 processors).

4.3 Visualization of wake relaxation downstream cutoff.

4.4 Influence of wake freezing on the integrated forces.

4.5 Visualization of two different wing panel resolutions.

4.6 Convergence behavior of time-averaged integrated force coefficients as a function of the spanwise panel resolution on the wing.

4.7 Visualization of two different propeller blade panel resolutions.

4.8 Convergence behavior of time-averaged integrated force coefficients as a function of the spanwise panel resolution on a propeller blade.

4.9 Visualization of the influence of the number of lifting lines used to represent the wing surface.

4.10 Convergence behavior of time-averaged integrated force coefficients as a function of the number of lifting lines on the wing.

4.11 Visualization of the influence of the time-step size used to model the system.

4.12 Convergence behavior of integrated force coefficients as a function of the number of time steps per revolution of the propeller.

5.1 HOFW model of the Küssner function.
5.2 Quasi-steady HOFW response to a sharp-edged gust as compared with the Küssner function. .................................................... 149
5.3 Unsteady DVE response to a sharp-edged gust as compared with the Küssner function. .................................................... 150
5.4 High resolution (30 lifting lines) HOFW response to a sharp-edged gust as compared with the Küssner function. .................. 152
5.5 Influence of damping of the $\partial \Gamma/\partial t$ term on the lift response. .... 153
5.6 A dimensioned diagram of the model used in the Dunsby et al. [9] experiment. ................................................................. 155
5.7 Variation of the time-averaged lift on the wing surface as a function of time-step number. .................................................... 157
5.8 Contour plot of pressure variation in the flow-field at a single moment in time as calculated using fully unsteady CFD. ............ 158
5.9 Comparison of the time-accurate lift predictions with CFD as with varying numbers of lifting lines and time-step sizes. .......... 159
5.10 Comparison of the frequency response of quasi-steady lift predictions with varying numbers of lifting lines and time-step sizes. .. 160
5.11 Comparison of the frequency response of unsteady lift predictions with varying numbers of lifting lines and time-step sizes. .... 161

6.1 The geometry of propeller 5868-9 in terms of non-dimensional section thickness (t/c), non-dimensional chord distribution (c/D), and pitch distribution (p/D) [10]....................................................... 166
6.2 A comparison of the power coefficients of the P5868-9 propeller as a function of advance ratio as predicted using the HOFW method and experimentally derived [10]..................................................... 167
6.3 A comparison of the propulsive efficiency of the P5868-9 propeller as a function of advance ratio as predicted using the HOFW method and experimentally derived [10]..................................................... 168
6.4 Geometry of JVX Proprotor in terms of non-dimensional section thickness (t/c), non-dimensional chord distribution (c/R), and section twist in degrees [11]. ......................................................... 170
6.5 A comparison of the predicted via the HOFW method and CAMRAD II [12], and experimentally derived [12] JVX Proprotor performance. ................................................................. 171
6.6 A comparison of the lift factor, $\lambda$, calculated using the HOFW method and that suggested by Smelt and Davies [13] ............. 173
6.7 A comparison of the change in lift coefficient due to the propeller slipstream as calculated using the HOFW method and the method suggested by Smelt and Davies [13] ............................. 174

xiii
6.8 Definition of angles and lengths relevant to jet flap analysis [14] (reprinted with permission)................. 175
6.9 A comparison of the change in lift coefficient due to the propeller slipstream as calculated using the HOFW method, the method recommended by Smelt and Davies [13], and the jet-flap method recommended by McCormick [14].................................................. 177
6.10 A comparison of the lift coefficient calculated as a function of angle of attack for the Dunsby experiment [9] as calculated using unsteady CFD, the HOFW method, the method recommended by Smelt and Davies [13], and the jet-flap method recommended by McCormick [14]. 180
6.11 A comparison of the spanwise lift distribution for the Dunsby experiment [9] at zero angle of attack as calculated using unsteady CFD and the HOFW method................................................. 181
6.12 A comparison of the drag coefficient calculated as a function of angle of attack for the Dunsby experiment [9] as calculated using unsteady CFD and the HOFW method................................................. 182
6.13 A comparison of the spanwise drag distribution for the Dunsby experiment [9] at zero angle of attack as calculated using unsteady CFD and the HOFW method................................................. 183
6.14 A comparison of the thrust as a function of angle of attack as measured in the Dunsby experiment [9] and with the HOFW method and CFD................................................................. 184
6.15 A comparison of the torque as a function of angle of attack as calculated in the Dunsby experiment [9] and with the HOFW method and CFD................................................................. 186
6.16 Photo of the experimental setup and geometry (in mm) of the model used by Veldhuis to investigate the influence of propeller position on downstream wing performance [3] (reprinted with permission). 188
6.17 A comparison of the lift-to-drag ratio of a propeller-wing system at a constant angle of attack with varying spanwise location of the propeller as calculated using the HOFW method and as determined experimentally by Veldhuis [3]. The propellers used in the two cases were different, presumably resulting in the difference in magnitude between the trends................................................................. 190
6.18 A comparison of the difference in drag due to a switch from up-outboard to up-inboard rotation orientation as a function of the lift coefficient as calculated using the HOFW method and as determined experimentally by Veldhuis [3]. The propellers used in the two cases were different, presumably resulting in the difference in magnitude between the trends................................................................. 191
7.1 Three view of the LCTR2-02 [15].

7.2 Baseline geometry for a new planar wing (top) and wing with 10% winglet (bottom) for the LCTR2.

7.3 The ideal loading of the LCTR2 planar wing with and without the proprotor.

7.4 Twist distribution for ideal loading of the LCTR2 planar wing.

7.5 The ideal loading of LCTR2 wing with a winglet.

7.6 Twist distribution for ideal loading of wing with winglet.

7.7 Comparison of the streamlines about a horizontal slice of the winglet (geometrically identified in blue in the figure on the right) as predicted with fixed- and relaxed-wake analysis for a single time-step.

7.8 Convergence of the CMAES algorithm for identification of a minimum drag condition for the planar LCTR2 wing.

7.9 CMAES optimized loading of the planar LCTR2 wing with and without profile drag.

7.10 The HEIST configuration as modeled within the HOFW method.

7.11 A summary of the rotation directions for each design case where a positive one (in yellow) indicates up-inboard rotation and a negative one (in blue) indicates up-outboard rotation. The most inboard propeller (closest to the wing root) is described in the left-most column progressing outboard with columns to the right.

7.12 A comparison of the drag polars of each of the six design cases described in Fig. 7.11.

7.13 A comparison of the net non-dimensional force in the drag direction as a function of the net non-dimensional force in the lift direction of each of the six design cases described in Fig. 7.11.

7.14 A comparison of the average propeller power coefficient as a function of the net non-dimensional force in the lift direction of each of the six design cases described in Fig. 7.11.

7.15 A comparison of the drag polars of each of the seven propeller vertical axis locations considered.

7.16 A comparison of the net non-dimensional force in the drag direction as a function of the net non-dimensional force in the lift direction for each of the seven propeller vertical axis locations considered. In this comparison, the propellers are kept at constant operating conditions.

7.17 A comparison of the average propeller power coefficient as a function of the net non-dimensional force in the lift direction for each of the seven propeller vertical axis locations considered.
7.18 An example of the processed used to determine the trimmed average propeller power coefficient for a set lift and $C_x$ of a given HEIST wing design. .................................................. 223
7.19 Trimmed average propeller power coefficient as a function of vertical location of propellers. .................................................. 224

B.1 Angle and coordinate system definitions for deriving the unsteady Kutta-Joukowski theorem. .................................................. 238

C.1 The control volume for Trefftz plane analysis of a single wing. . . . 244
C.2 The wake of a single planar wing shown in the Trefftz plane. . . . 245
C.3 Vortex sheet potential function formulation. ............................... 247
C.4 The wake of two parallel planar lifting surfaces as shown in the Trefftz plane. .................................................. 249
C.5 The intersecting wakes of two planar lifting surfaces as shown in the Trefftz plane. .................................................. 252

D.1 Control volume containing propeller and wing. .......................... 256

E.1 Graphical depiction of Munk’s third theorem. ............................ 264
E.2 Relationship between induced tangential velocity and surface normal velocity. .................................................. 267
List of Symbols

\(A, B, C\)  Circulation coefficients
\(AR\)  Aspect ratio \(\left(\frac{b}{S}\right)\)
\(b\)  Wing span
\(c\)  Chord
\(c_c\)  leading-edge suction coefficient
\(c_d\)  Two-dimensional drag coefficient
\(c_{d,0}\)  Two-dimensional viscous drag
\(c_{d,p}\)  Two-dimensional pressure drag
\(C_D\)  Three-dimensional drag coefficient
\(c_l\)  Two-dimensional lift coefficient
\(c_{l,max}\)  Maximum two-dimensional lift coefficient
\(C_L\)  Three-dimensional lift coefficient
\(c_n\)  Two-dimensional normal force
\(C_P\)  Power coefficient \(\left(\frac{P}{\rho n^4 D^4} \right)\) for propeller convention, \(\frac{P}{\rho A(\Omega R)^3}\) for rotor convention
\(C_T\)  Thrust coefficient \(\left(\frac{T}{\rho n^2 D^2} \right)\) for propeller convention, \(\frac{T}{\rho A(\Omega R)^2}\) for rotor convention
\(C_\mu\)  Momentum coefficient
\(DVE\)  Distributed vorticity element
\(D\)  Propeller diameter
\(\bar{F}\)  Body force vector
\(\dot{h}\)  Vertical kinematic velocity of trailing edge in cases of pitching and plunging
\(J\)  Advance ratio by propeller convention \((V/nD)\)
\(L/D\)  Lift to drag ratio
\(m\)  Number of lifting lines
\(M_\infty\)  Free-stream Mach number
\(n\)  Propeller rotational speed in revolutions per second
\(\bar{n}\) Normal vector
\(N\) Number of propellers
\(p\) Pressure
\(\frac{\rho}{\rho}\) Propeller pitch to propeller diameter ratio
\(P\) Position of quarter-chord location in the global reference frame
\(P_l, P_u\) Pressure on the lower and upper surfaces, respectively
\(q\) Free-stream dynamic pressure
\(r\) Propeller element radius
\(R\) Propeller radius
\(s\) Semi-chord \((c/2)\)
\(S\) Wing area
\(\overline{S}\) Strain-rate tensor
\(SDVE\) Surface DVE
\(T\) Thrust
\(\frac{T_A}{\bar{c}}\) Disk loading of propeller (ratio of thrust to disk area)
\(u, v, w\) Components of the velocity in the global reference frame
\(\bar{u}, \bar{u}\) Mean velocity, periodic-unsteady velocity
\(\bar{V}\) Velocity vector
\(V_{i,r}\) Real velocity of \(i^{th}\) element of propeller blade
\(V_{i,d}\) Discretized velocity of \(i^{th}\) element of propeller blade
\(V, V_\infty, U_\infty\) Free-stream velocity
\(w_a, w_1\) Velocity induced by the propeller in the axial direction
\(w_t\) Velocity induced by the propeller in the tangential direction
\(x\) Streamwise location
\(x_P\) Location of the propeller radius with respect to the leading edge of the wing
\(y\) Spanwise location
\(\alpha\) Angle of attack
\(\beta\) Propeller pitch angle
\(\delta\) Identity matrix
\(\Delta x_w/\Delta x_c\) Ratio of the distance traversed by a wing in a time-step to the length of a surface DVE
\(\gamma\) Vorticity
\(\Gamma\) Circulation
\(\epsilon\) Incidence angle in the body-fixed reference frame
\(\zeta\) Coordinate in DVE reference frame orthogonal to \(\xi\) and \(\eta\)
\(\zeta^*\) Second viscosity
\(\eta\) Spanwise coordinate in the DVE reference frame, propulsive efficiency \((\frac{C_T J}{C_P})\), correction for lack of full leading-edge suction attained in viscous flows

xviii
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\theta$</td>
<td>Collective pitch angle of the propeller</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Lift factor</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Advance ratio by rotor convention ($\mu = \frac{V_\infty}{\Omega R}$), fluid viscosity</td>
</tr>
<tr>
<td>$\xi$</td>
<td>Streamwise coordinate in the DVE reference frame</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Air density</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Rotor solidity</td>
</tr>
<tr>
<td>$\Upsilon$</td>
<td>Conservative body force</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Velocity potential</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Angular velocity of propeller, frequency of oscillations in unsteady flow</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>Angular velocity of proprotor</td>
</tr>
</tbody>
</table>
Acknowledgments

In the process of earning this doctorate, I have been supported by many members of the department, friends, family, students, and colleagues to whom I am truly grateful.

First and foremost, I would like to thank my advisor and committee members, Dr. Mark Maughmer, Dr. Edward Smith, Dr. Sven Schmitz, Dr. Götz Bramesfeld, and Dr. Gülen Kremer, for their technical expertise, patience, and support during this process.

I would additionally like to thank Dr. Mike Kinzel, Dr. Barnes McCormick, Dr. George Lesieutre, and Dr. Rob Kunz, all of whom were instrumental in the successful completion of my doctorate.

The contributions of the following people are also gratefully acknowledged: Dr. Van Cyr, Dr. Jim Coder, Ms. Margalit Goldschmidt, Mr. Steve Willits, Dr. Arnie Fontaine, Dr. Rhett Jeffries, Dr. Mike Krane, Dr. Laura Beninati, Dr. Nate Siegel, Dr. Katelyn Allers, Dr. Jean Lamont, Dr. Conor Marr, Dr. Charlotte DeVries, Dr. Michael Patterson, Mr. Frank Kody, Mr. Amandeep Premi, and Mr. Bernardo Vieira.

I am also grateful to my students both at Penn State and at Bucknell, for teaching me more than I could possibly have taught them.

Finally, I would like to thank my family and friends for their support. I could not have done it without you.
Introduction and Background

1.1 Motivation

An analysis method for the aerodynamic performance prediction of propeller-wing systems is desirable for many design applications. In the rotorcraft community, there is interest in continued improvement in tiltrotor cruise efficiency for both military and civil applications [15, 16]. For subsonic transport aircraft, it has been proposed that significant fuel savings can be achieved through high efficiency open-rotor concepts and advanced turboprops [17]. The use of multiple small-scale electric motors powering tractor propellers distributed across the span of a fixed-wing aircraft is being investigated as a means for drag reduction [18]. Another potential application can be found in the rapidly expanding field of unmanned aerial vehicles (UAVs), which are often propeller driven. One particular aircraft of interest within this field is the high-altitude long endurance (HALE) UAV, such as the AeroVironment Global Observer and the DARPA-sponsored Vulture [19], which feature propellers distributed across flexible, high-aspect-ratio wings.

All of these aircraft feature prominent aerodynamic interactions between pro-
propellers and wings. Consequently, the interaction should be considered early in the design process to ensure a thorough exploration of the design space. Within the traditional conceptual design phase, however, the design of propellers and wings occur separately, using either empirical or simple analytical methods to account for installation effects [20, 21]. Physics-based modeling of the interaction between the components is conducted within the preliminary or detailed design phases. Because many aspects of the design have been frozen at this point, there is less design freedom over which to explore potential beneficial interactions.

For conventional propeller-wing geometries, the use of empirical models may be adequate because the design space lies within the range of the historical data. With more complex and non-conventional designs, such as those previously noted, it becomes increasingly of concern that these models may fail to capture important aspects of the interaction. For example, there may be a difference in torque between an isolated propeller and one operating near a wing or other propellers. Even if this difference is small, an aircraft featuring many propellers distributed along the span may perform significantly better or worse at the system level when this effect is taken into account. An example on the component level is that a winglet on a tiltrotor may increase or decrease the drag of the wing depending on the accuracy of the model used to design it. This is a result of the sensitivity of the effectiveness of the winglet to the incidence angle of winglet root, referred to as the toe angle [22]. If the winglet experiences a change in angle of attack due to the wake of the proprotor from that expected, a performance penalty rather than a benefit can result.
1.2 Structure of Thesis

The objective of this thesis is to describe the development of an aerodynamic model to enable conceptual design-space exploration for vehicles of complex geometry featuring significant propeller-wing interaction. To complete this objective, it is first necessary to understand the physics of the interaction between a propeller and wing to ensure that all relevant parameters have been taken into account. This is addressed in Chapter 1, along with a description of the historical and currently available methods in the context of their applicability to the problem at hand. The method for determining the full flow field is then described in Chapter 2, and the approach to calculating lift, drag, thrust, and torque from this solution is detailed in Chapter 3. The influence of grid resolution on performance predictions is addressed in Chapter 4. The time accuracy of the method is then specifically addressed in Chapter 5 through comparison with an analytical solution and unsteady CFD. In Chapter 6, the time-averaged integrated forces determined using the method are compared with experimental data, CFD, and two accepted semi-empirical methods. The method is applied in two design studies in Chapter 7. Finally, conclusions and future work are provided in Chapter 8.

1.3 Background

The propeller and wing induce velocities on one another, potentially altering the performance of each component [3, 4, 16, 23, 24]. The mechanisms that result in the change in performance for each component and the parameters (both geometrical and operational) that influence these mechanisms must be explored to ensure that they are captured by the new model. In addition, quantification of the signifi-
cance of interference effects provides justification for the additional computational expense of moving beyond empirical models in conceptual design.

1.3.1 Influence of the Propeller on the Wing

1.3.1.1 The Wake of the Propeller

As an upstream propeller produces thrust, it induces velocities within its wake (referred to as the propeller slipstream) that impinge upon the wing downstream. These velocities have components in the axial, tangential, and radial directions, as defined in Fig. 1.1. When thrust is produced, the induced tangential velocity is in the same direction as the blade rotation. The radially induced velocities are typically negligible in magnitude and are therefore excluded from this discussion.

Figure 1.1: Definition of axial, tangential, and radial induced velocities due to a propeller.

To determine the impact of the induced velocities on the wing performance, it is appropriate to begin by considering the following questions:

- What are the magnitudes of these velocities relative to the freestream? On what do these magnitudes depend?
• How do the velocities vary with radial location?

• How do the velocities vary in the streamwise direction?

• Do the velocities induced within the slipstream vary with time?

The average magnitude of the axial induced velocity, $w_a$, in the propeller plane depends primarily on the ratio of thrust ($T$) to disk area ($A$), referred to as disk loading, and can be approximated according to McCormick [25] as

$$w_a = \frac{1}{2} \left[ -V_\infty + \sqrt{V_\infty^2 + \frac{2}{\rho} \frac{T}{A}} \right]$$

(1.1)

where $\rho$ is the air density. This relationship is based on momentum theory and is shown in terms of the induced axial velocity as a function of disk loading for a range of freestream velocities in Fig. 1.2. At a given velocity and air density, the average induced axial velocity increases with increasing disk loading. At a given disk loading, the average induced axial velocity decreases with increasing velocity. If disk loading and freestream velocity are held constant, a reduction in air density increases the induced axial velocity.
Figure 1.2: Variation of induced axial velocity as a function of disk loading as defined by Eq. 1.1 at sea level density for varying freestream velocities. The two aircraft examples are provided for context and are calculated at their respective cruise conditions.

To provide context to the trends shown in Fig. 1.2, the estimated induced axial velocity of a tiltrotor in cruise and a general aviation (GA) aircraft in cruise are included. The tiltrotor disk loading \( T/A \approx 2.4 \) and cruise velocity \( V_\infty \approx 230 \) kts were determined from wind-tunnel test conditions of the JVX proprotor [12], which is very similar in design to a V-22 Osprey proprotor. The cruise altitude for the tiltrotor case was assumed to be 15,000 ft. The general aviation aircraft disk loading \( T/A \approx 10.4 \), airspeed \( V_\infty \approx 117 \) kts, and cruise density \( \rho = 0.00211 \) slugs/ft\(^3\) were taken from Piper Cherokee data found in McCormick [14].

The average magnitude of the tangential induced velocity within a propeller wake depends on the torque of the propeller, and thus depends on its geometry and operating conditions [26]. Because an increase in thrust typically correlates
with an increase in torque for a given propeller, it also results in an increase in induced tangential velocity. As an example, a propeller designed for the operating conditions of a GA aircraft in a sea-level cruise (2400 RPM, 187 ft/sec, and 310 lbs of thrust) was analyzed using QPROP [1] over a sweep of pitch angles at three velocities (the same as those considered in Fig. 1.2). QPROP is an extended blade-element vortex method that accounts for both profile drag and radially-varying self induction velocities. The resulting induced tangential velocities as a function of disk loading are shown in Fig. 1.3. These tangential velocity results are shown as an example only, as unlike the average induced axial velocities, the average induced tangential velocities vary with the blade design. This is to say that two propellers of the same radius operating at the same conditions but with different planforms may produce different average induced tangential velocities. Still, the trends are similar to those presented in the case of induced axial velocity in that increased loading results in increased tangential velocities.
The average variation of the axial induced velocity in the streamwise direction, based on the velocity induced along the axis of a semi-infinite helical vortex filament, can be approximated as

$$\frac{w_a(x)}{w_{a,0}} = 1 + \frac{x/R}{\sqrt{1 + (x/R)^2}}$$  \hspace{1cm} (1.2)$$

where $x$ is the streamwise distance from the propeller plane, $R$ is the radius of the propeller, and $w_{a,0}$ is the induced axial velocity in the propeller plane [14]. In the far wake (as $x$ approaches infinity), the induced axial velocity doubles from its value in the propeller plane. This far-field result is consistent with classical momentum theory [27,28].

The average variation of the tangential induced velocity in the streamwise di-
rection can be approximated using a vortex model of the propeller (such as that shown in Fig. 1.4) with a step function, where upstream of the propeller there are no induced velocities, and downstream the velocities are of constant magnitude. The lack of induced tangential velocities upstream of the propeller plane is a result of the influence of the bound vortex being equal and opposite to the influence of the free vortices [3, 26, 29]. As the distance downstream of the propeller plane increases, the influence of the bound vortex decreases and that of the trailing vortices increase. This results in constant disk-averaged induced tangential velocity with increasing downstream distance. This approach and the resulting explanation is attributable to Glauert [29] and was also addressed by Weick [26] and more recently by Veldhuis [3].

Figure 1.4: Illustration of a vortex-wake model of a propeller neglecting slipstream contraction.

There are also variations in the induced axial and tangential velocities along the span of the propeller disk. These variations depend on the design of the propeller
blade and its operating conditions. As an example, the two planforms shown in Fig. 1.5 were analyzed using QPROP [1]. The first propeller was designed for minimum induced losses using QMIL [1], which is a blade element momentum method, and is referred to as the MIL planform. The second propeller is identical in twist distribution, but uses a rectangular planform in which the chord was adjusted to match the thrust at the original operating conditions. The induced velocities within the propeller plane of the two propellers operating in the same conditions and at the same thrust are shown in Fig. 1.6. As shown in the figure, this example demonstrates the dependence of induced velocities on the propeller blade design.

Figure 1.5: Planforms of the two example propeller blades analyzed in Fig. 1.6.
Figure 1.6: A comparison of the induced velocities for two propellers of differing designs at the same thrust and advance ratio within the propeller plane.

As an example of the dependence of induced tangential and axial velocities on the operating conditions of a given propeller, the variation of the velocity profiles of the MIL planform at varying advance ratios is shown in Fig. 1.7. As the thrust coefficient increases (with reducing advance ratio), both the axial and tangential induced velocities increase in magnitude. In this case, the increase is primarily seen in the tip region for axial induced velocities and in the root region for tangential induced velocities, although this behavior is again expected to depend on the blade design.
Figure 1.7: An example of the variation of induced axial and tangential velocities with advance ratio within the propeller plane as calculated with QPROP [1].

Of course, the “non-averaged” velocities induced by a propeller vary at a location in the wake as a function of time. An imaginary point in a stationary global reference frame near the propeller plane within the wake will experience variations due to its proximity to each propeller blade and due to the vorticity shed from each blade. These variations have been investigated experimentally with Laser Doppler Velocimetry (LDV) [6, 30, 31] and with hot-wire anemometry [32]. Several data sets are reproduced from Lepicovsky [2] which demonstrate this variation. This experiment used a two-bladed model-airplane propeller (diameter of 330 mm) at 4250 RPM. Examples of the variation of axial and tangential induced velocities
with streamwise location are shown in Fig. 1.8 and Fig. 1.9. Examples of variation of the axial, tangential, and radial induced velocities with radial location are shown in Fig. 1.10.

Figure 1.8: Example of axial induced velocity variation at a single radial location [2] (reprinted with permission of the ASME).

Figure 1.9: Example of tangential induced velocity variation with axial location [2].
Figure 1.10: Example of induced velocity variations (left - axial, center - tangential, right - radial) with radial location [2].
1.3.1.2 Impact of the Propeller Wake on the Wing

Having characterized the wake of the propeller, it is now appropriate to consider the impact of that wake on the wing. Following from the discussion of the propeller wake, the questions of interest include:

- How does the propeller wake influence the wing performance?
- What parameters have an impact on this influence?
- What is the magnitude of this influence?
- Is the wing performance within the wake of the propeller time dependent?

The influence exerted on the wing by the propeller is a consequence of the velocities induced by the propeller that are encountered by the wing. For up-inboard rotation, the induced tangential velocity results in a higher angle of attack inboard and a lower angle of attack outboard of the propeller. If the propeller rotates in the opposite direction (down-inboard), the effects are reversed with decreased lift inboard and increased lift outboard. In addition, the induced axial velocity results in a higher dynamic pressure within the slipstream, increasing the local lift and drag. These two effects in combination can also significantly influence the spanwise lift distribution, an illustration of which is provided Fig. 1.11. One impact of this modified lift distribution is the potential for increased vortex-induced drag of the wing. The magnitude of the influence of the propeller on the wing depends on the operating conditions, rotation direction, location, orientation, and diameter of the propeller with respect to the wing. As the influence of the propeller operating conditions on the induced velocities within its wake have already been
addressed, the remaining parameters of propeller rotation direction and location, incidence, and diameter with respect to the wing will be the focus of this section.

Figure 1.11: Illustration of the impact of the propeller wake on the wing lift distribution as it depends on propeller rotation direction. Modified and reprinted with permission from Ref. [3].

The direction of the propeller rotation (and consequently the induced tangential velocity) has a clear impact on the spanwise lift distribution, which translates to changes in lift and vortex-induced drag. This effect is evident in the wind-tunnel test results shown in Fig. 1.12, in which the spanwise lift distribution of an
unswept, untwisted wing under the influence of an up-inboard rotating and an up-outboard rotating propeller is provided [3]. Up-inboard rotation has been shown to be preferable to down-inboard rotation in terms of lift-to-drag ratio ($L/D$), as it results in less of a departure from an elliptical lift distribution [3, 33, 34]. An example of the magnitude of this benefit can be found in the results of a three-dimensional wind tunnel test conducted on a model of a Fokker F-27 at Delft University of Technology [3]. In this case, up-inboard rotation results in a reduction of four counts of total aircraft drag over down-inboard rotation at a typical cruise lift coefficient of 0.4. Increased benefits were observed at higher lift coefficients (approximately 12 counts reduction at $C_L = 0.9$) and with inboard flap deflection (approximately 30 counts reduction at $C_L = 0.82$) [3].

![Figure 1.12: Wind tunnel test results showing the spanwise lift distribution of a straight, untwisted wing ($AR = 5.33, \alpha = 4^\circ$) with and without the influence of a propeller ($J=0.85, C_T = 0.168$) [3].](image)

Another factor influential in wing performance is the distance between the
propeller and wing. As is evident from the previous descriptions of a propeller wake, the tangential induced velocities will be nearly constant with distance aft of the propeller, while the axial induced velocity doubles in magnitude from the propeller plane to the far wake. The resulting impact on performance is that for a given thrust condition, the lift of the wing tends to increase with increasing distance between the wing surface and propeller plane. An example of this is shown in Fig. 1.13. The drag of the wing considered in this figure is fairly insensitive to the streamwise location of the propeller, although this is likely dependent on the specific design. In this case, an increase in profile drag is offset by a decrease in induced drag [4].

Figure 1.13: Change in lift coefficient with propeller streamwise location for a high-speed case (J=1.63, $C_T = 0.046$) and a low-speed case (J=1.00, $C_T = 0.251$) as calculated using a modified vortex-lattice method [4].

The spanwise location of the propeller also has a significant impact on wing performance. For example, in [3], the L/D of a wing at a moderate angle of
attack ($\alpha = 4.2^\circ$) with a single up-inboard rotating propeller ($J=0.92, C_T = 0.127$) was shown experimentally to increase from 18.7 when the propeller was at approximately 30% of the half span, to 26.8 when the propeller was located at the wing tip. This increase occurred mostly when the propeller was outboard of 80% of the semi-span, and was attributed by the author of the study to the tangential induced velocity from the propeller counteracting the wing tip-vortex, resulting in an increased effective aspect ratio. An illustration of this effect is shown in Fig. 1.14. Despite its potential, a wing-tip propeller is not practical in most cases due to structural and aeroelastic concerns.

![Figure 1.14: Illustration of the interaction between the wing-tip vortex and the wake of a propeller located at the wing tip. The propeller rotating up inboard produces tangential velocities that are opposite of those produced by the wing-tip vortex.](image)

Wing lift and drag can also be influenced by the vertical location of the propeller. This a result of the distribution of induced velocities across the diameter of the propeller. For example, if the propeller from Fig. 1.7 was placed 0.5 radii above the wing and the thrust vector was aligned with the free-stream, at that spanwise location the wing would experience an increase in effective free-stream velocity of approximately 25%. Alternatively, if the propeller axis were placed in-
plane with the wing, the free-stream velocity would remain relatively unaffected. The lift distributions resulting from these two designs would be notably different. In the case of the tangential induced velocity, the vertical location of the propeller does not only influence the magnitude of the induced velocities, but also the direction. For example, the tangential induced velocities of an axis in-plane with the wing would be entirely perpendicular to the span, whereas an offset axis would introduce spanwise components.

The influence of the vertical location of the propeller with respect to the wing was investigated experimentally by Veldhuis [3]. The model for this study consisted of a half-span wing (with reflection plane) and a separated and moveable propeller-nacelle unit. The variations in lift and drag coefficient of the wing as a function of propeller vertical location for two thrust conditions are shown in Fig. 1.15 as an example of the results of this study. The variation in induced velocities across the propeller wake are particularly evident in the high thrust results ($C_T = 0.985$). The maximum lift coefficient for the high thrust case occurs when the propeller hub is approximately one-half radius above the wing. The author of the study attributes this result to the wing surface operating in the maximum induced axial velocity of the propeller with favorable influences due to swirl effect and slipstream contraction. The latter two effects are then credited for the reduction in lift coefficients at negative propeller offsets.
Figure 1.15: Experimentally determined wing lift coefficient (upper) and drag coefficient (lower) as a function of propeller vertical location at a low and high thrust condition ($\alpha = 4^\circ$) [3].

In addition to the location of the propeller with respect to the wing, the propeller shaft angle relative to the wing can also be influential to wing performance. In [4], this effect was investigated experimentally using the separated propeller-nacelle/wing model previously described. Within conventional limits ($\pm 2^\circ$), the influence of the propeller incidence angle on the lift and drag of the wing was found to be fairly small. When larger angles were considered, downward incidence angles were shown to enhance the lift-to-drag ratio of wing significantly. For example, a propeller incidence angle of $-14^\circ$ increased the wing lift-to-drag ratio by approximately 9.6% at a wing angle of attack of 4.2º over that found at a propeller incidence angle of 0º. This effect was shown to be more potent at a wing angle of attack of 8.4º, resulting in an increase in lift-to-drag ratio of 23% over the baseline.
condition.

Another important propeller-wing interaction parameter is the propeller diameter relative to the wing semi-span, as this dictates how much of the wing is operating within the propeller slipstream. The most notable example of this is the tilt-wing configuration, where it is desirable for the entire wing to be engulfed in the propeller slipstream to reduce wing buffetting during transition between forward flight and hover operations [35]. Dunsby et al. [9] performed a wind-tunnel test on a tilt-wing, in which a single propeller with a diameter to wing semi-span ratio of 95% was used. Some representative results of this study are shown in Fig. 1.16 to quantify the magnitude of the interaction when nearly the entire span of the wing is enveloped by the propeller slipstream. Significant reductions in lift are evident at all three angles of attack when the propeller is operational due to down-inboard rotation.

Figure 1.16: Local circulation as a function of spanwise location for propeller off and down-inboard rotation ($C_T = 0.155$).
Having characterized the parameters which influence the interaction between the propeller and wing, it is relevant to consider the influence of the time-dependency within the propeller wake on the the wing surface pressure distributions, and consequently wing lift and drag. Johnston and Sullivan [5] investigated unsteady wing surface pressures in the wake of a propeller. Sample results of this experimental study are shown in Fig. 1.17, where the pressure coefficients are shown as a function of time (normalized by the time for a complete propeller revolution) at a single chordwise location near the leading edge. These fluctuations were shown to decrease in magnitude toward the trailing edge due to viscous effects along the wing. Although the study did not report aggregate changes in lift, clear variations in the upper and lower surface pressure distributions indicate the potential for time dependencies in integrated lift.

As further evidence of the time-dependent propeller-wing interaction, Roosenboom et al. [6] investigated the slipstream of an 8-bladed propeller (J=0.79) impinging upon a wing using Particle Image Velocimetry (PIV). Vorticity contours for a nonzero thrust case at a moderate angle of attack are shown in Fig. 1.18. These results verify the periodic nature of the flowfield. In a later study Roosenboom et al. [36] used an unsteady Reynolds-averaged Navier-Stokes (URANS) simulation to analyze the same case. The analysis showed good agreement with the experimental results. A time history of the lift, drag, and side force coefficients development within the URANS analysis showed periodic oscillations due to the interaction between the propeller and wing.
Figure 1.17: Surface pressure coefficients as a function of normalized time at a single chordwise location ($x/c=0.04$, $J=0.37$, wing angle of attack = $10^\circ$) [5] (reprinted with permission of AIAA).
Thom and Duraisamy [7] also investigated the unsteady nature of propeller-wing interactions computationally. In that study, the Euler equations were solved over high-resolution overset meshes. For comparison, the propeller was also modeled with a highly-accurate actuator disk based on the unsteady results for a time-averaged solution. As shown in the results reproduced in Fig. 1.19, the authors found up to 40% differences at certain locations along the span between time-averaged loads and unsteady loads. As a result, inclusion of uncertainty due to unsteady effects was recommended in the design of propeller-wing systems.

1.3.2 Influence of the Wing on the Propeller

The influence of the propeller on the wing is intuitive, as the propeller wake clearly intersects the wing surface. The influence of the wing on the propeller is less obvious, leading to the following questions:
Figure 1.19: Comparison of lift and inviscid drag distributions on a wing within a propeller wake for time-averaged and unsteady Euler analysis [7] (reprinted with the permission of AIAA).
• How does the wing influence the propeller performance?

• What parameters have an impact on this influence?

• What is the magnitude of this influence?

• Does a propeller, in the vicinity of a wing, have time-dependent performance?

There are two primary mechanisms for the influence of the wing on the propeller, both of which will be addressed in depth in subsequent paragraphs. The first mechanism is a reduction in the tangential velocity within the wake of the propeller, referred to as “swirl recovery”. The second mechanism is the asymmetric inflow that results from the upwash forward of the leading edge of the wing. For typical propeller-driven aircraft, swirl recovery is more dominant than asymmetric inflow conditions, although the magnitude of each effect depends on the propeller position, propeller operating conditions, and wing loading [3, 34].

A reduction in rotational velocity within the wake of a propeller increases propulsive efficiency because swirl within the wake does not contribute to the thrust of the propeller, and is thus an energy loss. As a result, any reduction in rotational energy improves the efficiency of the system. Physically, swirl recovery occurs when the velocity induced by an additional surface and/or its wake interacts with the propeller wake to reduce rotational velocities. For example, Veldhuis [3] attributes swirl recovery to a reduction in the helix angle of the propeller wake caused by the upwash forward of the wing and downwash aft of the wing. Kroo [34] approaches the phenomenon theoretically and asserts that through an extension of Munk’s stagger theorem [37], the streamwise positions of the wing and propeller are irrelevant for fixed circulation distributions on each component, as the losses either manifest as a change in thrust for the propeller or a change in
induced drag for the wing.

The asymmetric inflow mechanism due to the upwash forward of the wing is a consequence of the production of circulation on the wing that corresponds to lift generation. This upwash varies in magnitude with the wing loading as shown in Fig. 1.20, where a representative RANS CFD calculation performed by the author is presented. In cases where the propeller is close enough to the wing to be influenced by this upwash, either because of the physical location of the propeller or because of the wing loading conditions, the result is asymmetric inflow. If the propeller diameter is small compared to the wing span and located in a region where the upwash is nearly uniform, the increase in lift on the downward moving blade is partially compensated by the decrease in lift on the upward moving blade. Alternatively, a propeller of large diameter with respect to the wing would experience variations in upwash across the span. In the extreme case of a tiltrotor on which the nacelle is located at the wingtip, upwash is present for the inboard blade, but not on the outboard blade.
Figure 1.20: Induced upwash of an NACA 0012 airfoil \( V_\infty = 200 \text{ ft/s} \) at \( \alpha = 5^\circ \) (upper) and \( \alpha = 10^\circ \) (lower) as calculated using Solidworks® Flow Simulation.

In all of these cases, the propulsive efficiency can be impacted, which in turn, influences the performance of the propeller-wing system. As an example of the magnitude of this interaction, propeller efficiency calculated using a blade-element method [3] is plotted versus advance ratio in Fig.1.21, with and without the influence of the wing induced upwash. Because the propeller influences the lift of
the wing as well, it follows that a true assessment of the system efficiency would require iterative trim of thrust and lift to desired conditions.

Figure 1.21: Influence of wing upwash on propeller efficiency as calculated using a blade element method [3]. In this case, the propeller is located one radius in front of the leading edge of the wing \( x_P = 1.0R \).

1.3.3 Influence of Mutual Interaction on Aircraft Performance

As an indication of the design impact of the interactions described in this section, a propeller and wing were analyzed individually and then as a coupled system using the method developed in this thesis. The resulting power available and power required are provided in Fig. 1.22. For this case, the power available varied when interaction was accounted for by about 2% from the uncoupled analysis, whereas the power required varied by up to 20%. As verified by these results, propeller-wing interaction can influence both the power-on stall speed and the maximum
speed of an aircraft, as well as its rate of climb within the flight envelope.

To indicate the relative magnitude of the influence of the components of drag and propeller power and to give context to the potential gains possible through design for minimum induced losses, a drag buildup and a propeller power buildup were performed using the results of Kroo [34] and several empirical approaches [20, 21, 38]. The aircraft for this example was assumed to roughly resemble a conventional turboprop driven, medium range airliner. The drag build-up is shown in Fig. 1.23 and the propeller power build-up is shown in Fig. 1.24. These figures help to elucidate the importance of, for example, including profile drag in an assessment of a propeller-wing design. They also put in context the relatively small gains available in designing for minimum induced losses in propeller wing interaction for a conventional aircraft.
Figure 1.22: Influence of mutual interaction on power available and power required of an example aircraft.
Figure 1.23: Example drag build-up for a conventional turboprop aircraft estimated using empirical methods.
Figure 1.24: Example power build-up for a conventional turboprop aircraft estimated using empirical methods.

1.3.4 Summary

This section has provided an overview of the numerous interaction mechanisms that arise between a propeller and wing. The propeller design, location, orientation, and operating conditions have been shown to be influential to the wing performance as a result of induced velocities. The most important system parameter in determining the magnitude of the induced velocities is the propeller disk loading, as increased disk loading results in increased induced velocities. The design and operating conditions of the propeller significantly influence the distribution of induced velocities across the propeller disk, which in turn, impacts the performance of the wing as well. Because this induced velocity distribution is not uniform across
the propeller plane, nor in the downstream direction, the location of the wing with respect to the propeller also influences the performance of the wing. Finally, time dependencies within the propeller wake translate to time-dependent lift and drag production by the wing. The wing has also been shown to impact the performance of the propeller. The spanwise lift distribution on the wing results in an upwash distribution forward of the wing that affects the propeller inflow. The magnitude of this influence depends on the wing loading and the diameter of the propeller with respect to the span of the wing. Because of the mutual interaction present in propeller-wing systems and the many design parameters that influence it, it can be concluded that in most cases it is necessary to account for propeller-wing interaction within the conceptual design process.

1.4 State of the Art

The ideal characteristics of a method for calculating the aerodynamic performance of propeller-wing systems for conceptual design applications include four main attributes. First, the method must be capable of modeling both wings and propellers at a geometric design level (i.e. a change in propeller blade shape can be translated into a change in thrust) and of capturing the mutual interaction between the two systems, including relaxed-wake (as will be explained later in the chapter) and time-dependent effects. The second attribute is that the method must be accurate. Accuracy here does not necessarily indicate that the method must match experimental data exactly. What is more important in the context of design is that the trends in predicted performance with varying design parameters are accurate and reliable. The third attribute is that the method must demonstrate computational efficiency, both in terms of time and memory. This enables the large scale
design exploration required to identify local optima. The fourth attribute is that the method must be numerically robust in that it must be able to handle multiple interacting wakes without solution divergence and without heavy reliance on solid-core models or other empirical corrections.

Consideration of previous modeling approaches in the context of these requirements is warranted. McCormick [14] recommends the semi-empirical methods of Smelt and Davies [13] and Kuhn [39]. The details of these methods are discussed extensively in Chapter 6, where they are used for comparison with the method developed in this thesis. These methods predict several important trends at low computational cost. They also empirically account for many effects neglected by other low order methods, such as viscosity and separation. Unfortunately, they are not necessarily appropriate for predictions outside of the limited set of experimental data on which they are based. In addition, neither method accounts for the influence of the wing on the propeller or the influence of the propeller geometry.

A thorough review of the potential flow modeling attempts pre-1975 was compiled by McVeigh et al. [40]. All of these approaches are based on potential flow theory, and thus assume inviscid and irrotational flow. The most simple model used was a lifting line immersed in a circular uniform axial jet without rotation. This approach is incapable of resolving the influence of the propeller blade design on the resulting inflow. An additional drawback of this approach is the inability of the lifting-line method to resolve the lift over the small span section of the wing immersed in the jet. The use of a lifting surface approach (with 3/4 chord flow tangency enforced) in place of the lifting line mitigated the latter concern, but was still unable to capture swirl effects or spanwise variations in induced velocities. Multiple slipstream tubes accounted for the spanwise variation in induced velocities, but
remained incapable of capturing swirl effects. McVeigh et al. [40] used a converged blade element momentum theory analysis of the propeller to determine induced velocities (both axial and tangential) and then applied the calculated velocities to a modified lifting-line model. A more recent extension of McVeigh’s approach is that of Stone [41], in which a fixed-wake panel method was combined with a converged blade element momentum theory analysis. In all of these approaches, mutual interaction between the propeller and wing was neglected.

Many potential flow methods developed within the past three decades are more sophisticated in their approach. In the models of Witkowski et al. [42] and Veldhuis [3], vortex-lattice methods (VLMs) were used to model both the propeller and the wing. Witkowski et al. [42] modeled the two systems within the same potential field, and accounted for viscous effects on the wing with strip theory. Veldhuis [3] iterated between a blade-element momentum solution for the propeller and a vortex-lattice solution for the wing. In this formulation, the induced velocities due to one component were calculated, and then imposed upon the other. In addition, viscous effects on both the wing and the propeller were accounted for via strip theory.

Drawbacks of these approaches exist in the form of limited accuracy and numerical stability. VLMs suffer from significant discretization errors, and even with a large number of panels are limited in accuracy for the prediction of induced drag [43]. Any method that uses discrete vortex filaments to model the upstream surface’s wake, including VLMs, must account for the potential numerical interaction that occurs when vortex filaments, which are singular in nature, are in close proximity to the control points that dictate local circulation on the downstream surface.
An additional effect that is neglected by the previously described VLMs is that of wake relaxation. Because the wake is not a physical surface, it cannot sustain a load. This dictates that all vorticity in the wake must either move with the local velocity or be oriented such that it is parallel to the local velocity. In the previously described methods, a small-disturbance approximation has been applied that assumes that the changes in velocity from the free-stream condition are small. Under this assumption, if the system is steady and the wake is aligned with the free-stream, it is force free [44]. This type of wake is often referred to as “drag-free”. In reality, this assumption does not hold, as is evidenced by the roll-up of the wing-tip vortices commonly seen aft of an aircraft wing. Modeling of the deformation of the wake such that it remains force-free is commonly referred to as “wake relaxation”.

The effect of the inclusion of wake relaxation on predictions of aerodynamic performance depend on the type of system and operating conditions. In some cases, such as that of a single lightly loaded planar wing, the drag-free wake is typically sufficient. There is precedent to indicate that the influence of wake relaxation may not be negligible in the determination of propeller-wing system performance, such as is the case for interacting wings and wakes [45] and in the prediction of wind turbine performance [46]. Conjecture aside, it is difficult to quantify whether wake relaxation is truly influential for such systems without extensive wind-tunnel tests or CFD for comparison with fixed-wake models, or an accurate relaxed-wake method with which to compare fixed-wake results.

Relaxed-wake methods such as CAMRAD II [47,48], VSAero [49], and the work of Marretta et al. [50,51] allow for the deformation of the wake via the convection of control points by local induced velocities. Once again, due to the singular nature
of vortex filaments, this approach is potentially numerically unstable in the sense that the control points in the wake can encounter singular velocities when in close proximity with vortex filaments. These two methods mitigate this problem by replacing the singularity at the center of the vortex filament with a finite-core model. The drawback of this approach is that finite-core models are empirical or semi-empirical in nature and can result in lift and induced-drag predictions that depend on the core model and cutoff distance applied.

While potential flow methods can account for viscous effects with strip theory, they all model the flow as inviscid and irrotational. Removal of the irrotational assumption in propeller-wing modeling requires the use of the Euler equations, the approach taken by Dang [52], Whitfield and Jameson [53] and Thom and Duraisamy [7]. To directly include viscous effects, it is necessary to model the system with the complete or reduced forms of the Navier-Stokes equations, the approach taken by Roosenboom et al. [36] and Gomariz-Sancha et al. [54]. Although they capture more of the physics of the propeller-wing interaction, these approaches have limitations with respect to conceptual design applications of complex geometries, as they require significant user time for mesh generation and post-processing [55] and additional computational resources to acquire a converged flow solution. For example, in a study by Roosenboom et al. [36], an unsteady RANS simulation was used to analyze the slipstream of an 8-bladed propeller impinging upon a wing with remarkable accuracy. To acquire these results required a mesh grid of 23 million nodes on the wing and 36 million nodes on the propeller. Even with current high performance computing technology and state of the art mesh generation, this approach is not well suited for use in conceptual design.

A summary of the methods described and their qualifications in terms of the
four ideal characteristics is provided in Table 1.1. Empirical and low-order potential flow methods are notably fast and can be numerically stable, but often lack several desired capabilities, including the ability to capture mutual interaction, wake-relaxation and time-dependent effects. They are also limited in accuracy due to discretization errors and some have the potential for numerical instability. Methods in the second category of potential-flow methods are more capable than the first, although some still suffer from discretization errors and rely on solid-core models. Euler and Navier-Stokes approaches do not suffer from these shortcomings but at a computational resource cost that precludes their use in conceptual design studies.

<table>
<thead>
<tr>
<th>Method Type</th>
<th>Examples</th>
<th>Capable</th>
<th>Fast</th>
<th>Accurate</th>
<th>Robust</th>
</tr>
</thead>
<tbody>
<tr>
<td>Semi-empirical methods, lower-order</td>
<td>McCormick [14], McVeigh et al. [40],</td>
<td>X</td>
<td>✓</td>
<td>X</td>
<td>?</td>
</tr>
<tr>
<td>potential-flow methods with</td>
<td>Witkowski et al. [42], Veldhuis [3]</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>prescribed wakes</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>relaxation</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Euler and Navier-Stokes</td>
<td>Euler [7, 52, 53], URANs [36, 54]</td>
<td>✓</td>
<td>X</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>methods</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 1.1: Summary of some representative methods for aerodynamic analysis of propeller-wing systems and their respective capabilities.
1.5 Research Objectives

As is clear from Table 1.1, there is a gap in the available methods for aerodynamic analysis in the conceptual design of propeller-wing systems. The overarching objective of this thesis is to develop an aerodynamic analysis method to fill that gap and to enable design space exploration for non-conventional propeller-wing systems. Based on the desired attributes identified in the previous section, four sub-objectives must be met to show that the method is well-suited for this application. These objectives are:

1. The method must be shown to be capable of predicting performance of both propellers and wings on a design level (i.e. a change in design propagates through to a change in performance). It must also be capable of capturing mutual interaction between propellers and wings, both because of the proximity of the lifting surfaces and because of the interaction between the wakes. For this latter effect, wake relaxation must be taken into account. Finally, it must be capable of capturing time-accurate oscillations in forces due to interactions between the propeller and wing.

2. The method must be shown to be fast, both in terms of defining a system geometry and from the perspective of the computational time and space required to find a converged flow solution and extract relevant performance parameters.

3. The method must be shown to be accurate, if not in reproducing experimental results in both magnitude and trends, then at least in trends. Included in this is the ability to predict the unsteady oscillations due to interaction with
the wing.

4. The method must be shown to be numerically robust. This is to say that it must be shown that the results do not depend heavily on addition of core models or on the discretization of the lifting surfaces (within reasonable bounds) and that the solution does not diverge due to interaction between wakes.

1.6 Overview of Proposed Method

The method proposed to meet these objectives is an adaptation of Bramesfeld’s higher-order, free-wake (HOFW) method [8] to the setting of propeller-wing systems. Bramesfeld [8] developed a relaxed-wake lifting surface method that uses distributed vorticity elements (DVEs) to model lifting surfaces and wakes. The DVE consists of quadratically varying circulation at the leading and trailing edge connected by a sheet of linearly varying vorticity in the spanwise direction. When these elements are placed adjacent to one another to form a lifting surface or wake, the vorticity distribution is piecewise continuous. The resulting induced velocities in the wake are finite everywhere other than at the wing-tips without requiring the use of finite-core models. The lack of singular velocities within the wake make the HOFW method well-suited for modeling complex geometries, specifically those with interacting wakes. The higher-order nature of the elements also allows for improved resolution over zeroth- or first-order methods with fewer panels, which reduces the computational cost of a given solution for the same level of accuracy. Bramesfeld applied this method to single fixed wings and fixed wings in formation. Later, the method was applied to rotating wings [45] and horizontal axis wind
turbines [46, 56].

In the current approach, DVEs are used to model propeller blade and wing surfaces and wakes within a single time-stepping solution, thus intrinsically accounting for mutual and time-dependent interactions between the systems. In addition to addressing the independent kinematics of fixed and rotating lifting surfaces in the same flow solution, several improvements have been made to expand the effectiveness of the HOFW method for this application. The number of time-steps needed for convergence of propeller performance has been reduced through the implementation of an additional velocity variable. The ability to track circulation shed into the wake and to calculate non-circulatory effects were added to further the methods’ ability to capture time-dependent effects. To allow for a more complete performance prediction of all components, strip theory was implemented to account for viscous effects. The ability to designate the number of lifting lines on each lifting surface individually was added to enable flexibility with respect to the number of lifting lines needed on the downstream wing surface to resolve the influence of the propeller wake. Finally, to reduce computational time, a wake “freezing” parameter was implemented, which allows for faster computation due to a split between a near wake and a far wake.

The HOFW method for this application was implemented within a robust framework for defining wings and propellers, from which it derives additional utility. This framework enables modeling of multiple propellers with specified blade numbers, advance ratios, rotation directions, and axis locations in three-dimensional space. It also allows for the definition of multiple non-rotating lifting surfaces and facilitates full span simulation of like-rotating or counter-rotating propellers, as well as half-span simulation with a symmetry condition.
The resulting approach is capable, fast, accurate, and robust for performance prediction of propeller-wing systems, and is thus an improvement to the state of the art in modeling these systems for design applications.
Chapter 2

Theoretical Derivation and Numerical Solution Method

2.1 Overview

In this chapter, the foundations of the HOFW method are described with careful identification and assessment of all assumptions required. First, an overview of the HOFW method is provided through a description of its historical development and relation to classical lifting-line theory. Next, the governing equations for the method are derived, both to ensure applicability to propeller-wing systems and for clarity with respect to the underlying assumptions. With the governing equations in mind, the building block of the method, the DVE, is described. The approach to using DVEs to model physical lifting surfaces and their wakes is then addressed. Once the lifting surfaces and wakes have been defined, a system of equations must be solved to determine the strength of the surface DVEs. To conclude the chapter, the process used to solve this system of equations is explained.
2.2 Historical Context

In this section, the HOFW method is described broadly in terms of its historical development and in relation to classical lifting-line theory. This explanation provides both context and a brief overview of the method prior to a detailed description of the theory and application, which appears in subsequent sections.

The HOFW method is an extension of the multiple lifting-line method of Horstmann [43]. To clarify what is meant by “multiple lifting-line method,” it is worthwhile to begin with classical lifting-line theory as mathematically formalized by Prandtl [57]. In this theory, an unswept, high aspect-ratio wing in steady flight is modeled within the potential flow domain. Omitting the details for brevity, the theory represents the wing as a single potential vortex filament, referred to as the “bound” vortex or “lifting line,” that is oriented along the span of the wing.

Because a vortex filament cannot begin, end, or change strength in the fluid, any changes in strength along the bound vortex must be “fed” into the wake, which in the classical theory is assumed to be planar and aligned with the free-stream velocity. The strength of the wake is constant in the streamwise direction and varies in the spanwise direction as dictated by the derivative of the spanwise circulation distribution, $d\Gamma/dy$. If the circulation distribution in the spanwise direction is represented by a smooth and continuous function, the wake is a continuous sheet of vorticity. In this case, there are no singularities in the wake except for those present at each wingtip. Alternatively, if the circulation distribution in the spanwise direction is represented by a non-smooth function (such as a step function), the discontinuities are shed into the wake as well and are typically represented with vortex filaments, for which the induced velocities are singular.
The function representing the circulation distribution must satisfy Prandtl’s integro-differential equation, which ensures that the lift distribution and resulting downwash due to the shed wake are consistent for each section along the span. While solutions exist (most notably the elliptical lift distribution), the lift distribution of an arbitrary wing is not known a priori. Many numerical approaches have been developed to solve this problem.

In Weissinger’s method [58] (also referred to as the “Extended Lifting-Line Method” [59]) for example, the lifting line is discretized into segments of constant strength. The strength of each segment is adjusted to achieve flow tangency at the three-quarter chord point of that section. Because of this, the circulation distribution is discretized into several step functions and the wake is comprised entirely of singular vortex filaments. This is acceptable with a sufficient number of panels and careful placement of control points for a single wing with a fixed wake, but proves problematic for wake relaxation and multiple wing cases due to the singular induced velocities within the wake.

To better account for the chordwise distribution of circulation, it is possible to model the wing with multiple lifting lines along the chord. This is what is meant by a “multiple lifting-line method”. If each of the lifting lines exhibits a smooth and continuous distribution of circulation, the wake remains a continuous sheet of vorticity as well. Again, because most lift distributions are not known in advance, numerical methods must be employed. The simplest solution is an extension of Weissinger’s method to multiple chordwise lifting lines. This is equivalent to a special case of a vortex-lattice method in which the bound-vortex segments are constrained in length, location, and orientation to form continuous lines (from
root to tip) in the spanwise direction. In these cases, the strength of each segment of each lifting line is adjusted to achieve flow tangency at a specified control point for that segment only. Because the strength is constant across each segment (as in the Weissinger method), the wake consists of discrete and singular vortex filaments.

Horstmann’s multiple lifting-line method is more complex than this simple solution. Rather than defining the strength of each segment of the lifting line as a constant, Horstmann defined it as a second-order polynomial. The magnitude and slope of the polynomial on the boundary of each segment is constrained to match that of the adjacent segments on either side. In this way, the distribution of circulation along each lifting line is piecewise smooth and continuous, resulting in a wake that consists of a continuous sheet of vorticity with linearly changing strength in the spanwise direction. Thus, the wake-induced velocities in Horstmann’s model are finite everywhere except at the wing tips. The major benefit of this approach over a fixed-wake vortex lattice method is that it allows for rapid numerical analysis of complex geometries, for which lower order methods would require considerably more panels for equivalent resolution.

Bramesfeld [8] expanded upon Horstmann’s work to allow for a relaxed wake by combining two sets of Horstmann’s vortex filaments and semi-infinite vortex sheets to create a single element referred to as a distributed vorticity element (DVE). As in Horstmann’s method, these elements form continuous sheets of vorticity and therefore are singularity free within the wake other than for the logarithmic singularity at the wing tips. Because of this unique property, DVEs eliminate many of the numerical instabilities that hinder vortex-filament methods in the modeling of force-free (relaxed) wakes. Thus, Bramesfeld’s HOFW method includes wake relaxation, and as a result, is ideal for modeling complex lifting surfaces and wakes,
as well as wake interactions.

The HOFW method has been used for many applications. Bramesfeld and Maughmer [8, 45, 60] used DVEs to model fixed-wing aerodynamics, both for individual aircraft and in formation flight. Later, Basom and Maughmer [46, 61] applied DVEs to horizontal axis wind turbines. The approach developed by Basom and Maughmer was then used by Maniaci and Maughmer in the design of winglets for wind turbines [56, 62].

In this thesis, DVEs are used to model propeller-wing systems in the HOFW method. This application present a unique challenge, both because of the unsteady character of the flow-field and because of the inclusion of both fixed and rotating lifting surfaces. Accounting for unsteadiness in particular requires a reexamination of the theoretical foundations of the method, as it eliminates fundamental assumptions applied in previous applications. To ensure adequate scrutiny in light of this distinctive application, the method is derived from first principles in the subsequent section.

2.3 Theoretical Formulation of Method

In this section, the theoretical formulation of the HOFW method is addressed, first through derivation of the governing equations and then through description of the DVE. Specifically novel in this formulation is the inclusion of unsteady effects, which have been previously neglected but are present in propeller-wing systems.
2.3.1 Derivation of Governing Equations for Unsteady Potential Flow

In this section, the two governing equations for unsteady potential flow, the Laplace equation and the unsteady Bernoulli equation, are derived. The former is a result of continuity and the kinematic condition of irrotationality and is necessary to define a velocity field. The latter is used to convert this velocity field into a pressure distribution, which is necessary for the calculation of forces.

The unsteady Bernoulli equation is derived from the general Navier-Stokes equations through the successive application of assumptions. Each assumption is addressed in terms of its implications for propeller-wing systems prior to application. In the course of this derivation, the Laplace equation is derived as a means of mathematically describing a velocity field that obeys the assumptions applied to arrive at the unsteady Bernoulli equation. These derivations follow the nomenclature and approach found in Wilcox [63], but similar derivations can be found in most modern fluid dynamics textbooks.

The Navier-Stokes equations are a set of nonlinear partial differential equations that govern the flow of viscous fluids. There are two assumptions inherent in these equations. The first assumption is the continuum approximation, which requires that the fluid acts as a continuous substance rather than a set of discrete particles. This assumption is valid as long as a “fluid element”, as defined on the scale of interest for the flow being considered, contains enough molecules to allow for the identification of a valid statistical average of molecular motion [63]. At the scale of flows relevant for Earth-bound propeller-wing systems, this assumption is valid. The second assumption inherent in the Navier-Stokes equations is in the form
of the viscous stress tensor. Because the fluid is later assumed to be inviscid in this derivation, the terms influenced by the form of the viscous stress tensor are irrelevant as they are ultimately eliminated.

The Navier-Stokes equations in vector form are

\[
\rho \frac{D\mathbf{V}}{Dt} = -\nabla p + \rho \mathbf{F} + \nabla \cdot \left( 2\mu \mathbf{S} + \zeta^* \nabla \cdot \mathbf{V} \right) \tag{2.1}
\]

where \( \rho \) is the fluid density, \( \frac{D\mathbf{V}}{Dt} \) is the material (Eulerian) derivative of velocity with respect to time, \( p \) is the pressure, \( \mathbf{F} \) is any applied body force (such as gravity), \( \mu \) is the fluid viscosity, \( \mathbf{S} \) is the strain-rate tensor, \( \zeta^* \) is the second viscosity of the fluid, and \( \delta \) is the 3x3 identity matrix.

In an external flow, the influence of viscosity is felt primarily within a thin boundary layer on the surface of any solid object in the flow and within wake regions. Since the pressure is nearly uniform across this thin boundary layer (perpendicular to the surface outward), and the lift force is a result of the pressure differential between the upper and lower surfaces of a wing, lift can be accurately approximated using an inviscid solution as long as the circulation of the lifting surface is set such that the flow field is continuous. This is particularly of concern at the sharp trailing edge of an airfoil, where an instantaneous change in velocity resulting in an infinite acceleration of the flow could exist for an inviscid, incompressible solution, but would not exist in reality due to viscous and compressibility effects. The standard solution to this problem in an inviscid solution is to force smooth flow off the trailing edge of the lifting surface. This boundary condition, referred to as the Kutta condition, allows for a realistic prediction of lift when there is no separation.

Although lift can be well predicted with inviscid analysis, viscous effects can
not be neglected for the purposes of drag prediction. If the lift distribution is well predicted, however, viscous effects can be approximated on a section by section basis and then integrated over the span [64]. In this approach, the lifting surface is split into small two-dimensional strips over which the lift coefficient is assumed to be constant. The drag coefficient is then estimated using a look-up table at the lift coefficient, Reynolds number, and Mach number (if compressibility is of concern) of the two-dimensional section. It is assumed in this approach that the sections are independent of one another, which requires no flow in the spanwise direction. In reality, cross-flow does exist, although it is of limited influence for single wing cases except when they exhibit rapid changes in section, chord, sweep, and twist or at high angles of attack approaching stall conditions [64]. Propeller-wing interaction may induce cross-flow, but should be limited if the axis of rotation of the propeller is coincident (or close) with the plane of the lifting surface. As the axis of rotation of the propeller moves away from the plane of the lifting surface (in the z-direction), the induced cross-flow on the surface varies, with the potential to increase significantly depending on the location and operating conditions of the propeller (see Fig. 1.7). This assumption thus limits the applicability of the method for analysis of such cases.

Provided that the Kutta condition is enforced within the method and viscous drag is accounted for as a post-processing step as described, it is appropriate to eliminate viscous effects from Eq. 2.1 to obtain Euler’s equation,

$$\rho \frac{D\nabla}{Dt} = -\nabla p + \rho \mathbf{F}$$

(2.2)

Expanding the material derivative (left-hand side), Eq. 2.2 becomes
\[ \rho \left( \frac{\partial \vec{V}}{\partial t} + \vec{V} \cdot \nabla \vec{V} \right) = -\nabla p + \rho \vec{F} \]  
(2.3)

and expanding the left hand side further via a vector identity results in

\[ \rho \left( \frac{\partial \vec{V}}{\partial t} + \nabla \left( \frac{1}{2} \vec{V} \cdot \vec{V} \right) - \vec{V} \times \left( \nabla \times \vec{V} \right) \right) = -\nabla p + \rho \vec{F} \]  
(2.4)

A further simplification to Eq. 2.4 can be achieved by assuming the flowfield is irrotational. In the absence of curved shocks, heat addition, and viscous effects, a flowfield that is initially irrotational remains irrotational. Therefore, assuming low-speed flight and neglecting thermal effects, the flowfield can be assumed to be irrotational outside of the boundary layer. The irrotationality condition is given by

\[ \nabla \times \vec{V} = 0 \]  
(2.5)

which allows Eq. 2.4 to be reduced to

\[ \rho \left( \frac{\partial \vec{V}}{\partial t} + \nabla \left( \frac{1}{2} \vec{V} \cdot \vec{V} \right) \right) = -\nabla p + \rho \vec{F} \]  
(2.6)

It is desirable to regroup the terms in Eq. 2.6 in a more usable manner. To do so, it is helpful to assume that the fluid density is constant with changes in pressure, i.e. the flow is incompressible. This assumption is also essential in the formulation of the potential function to be described later. For the flow to be considered incompressible, the Mach number of the system modeled should be less than approximately 0.3 everywhere, including all locations on the lifting surfaces [63]. This limits applicability of the method significantly, specifically with regard to the tip-speed of propellers. One mitigating factor for this limitation is
that for higher Mach numbers that are still subsonic, it is possible to implement a small-disturbance compressibility correction to an incompressible solution. This correction and its implementation are discussed in Section 2.4.3.

If the applied body force is assumed to be a conservative force, given by \( \mathbf{F} = -\nabla \Upsilon \), then with the addition of the incompressibility assumption, the terms in Eq. 2.6 can be rearranged as

\[
\frac{\partial \mathbf{V}}{\partial t} + \nabla \left( p + \frac{1}{2} \mathbf{V} \cdot \mathbf{V} + \Upsilon \right) = 0 \quad (2.7)
\]

Conservation of mass additionally requires:

\[
\nabla \cdot \mathbf{V} = 0 \quad (2.8)
\]

If the function, \( \phi \), is defined such that

\[
\mathbf{V} = \nabla \phi \quad (2.9)
\]

that function automatically satisfies the irrotationality condition defined in Eq. 2.5 (since \( \nabla \times \nabla \phi = 0 \)). \( \phi \) is referred to as the velocity potential. Combining Eq. 2.8 and 2.9 gives

\[
\nabla^2 \phi = 0 \quad (2.10)
\]

which is to say that the velocity potential satisfies Laplace’s equation. Because Laplace’s equation is a linear partial differential equation, it obeys the superposition principle. This principle indicates that if two functions, each of which satisfies the Laplace equation individually, are added to form a third function, this third function will satisfy the Laplace equation as well. This is an important character-
istic of potential flow, as solutions to complex flows can be built by adding many potential flow solutions.

If $\nabla \phi$ is inserted into Eq. 2.7 in place of the velocity, Eq. 2.7 becomes

$$\nabla \left( \frac{\partial \phi}{\partial t} + \frac{p}{\rho} + \frac{1}{2} |\nabla \phi|^2 + \Upsilon \right) = 0$$  \hspace{1cm} (2.11)

Eq. 2.11 can be integrated in space to yield the unsteady Bernoulli equation,

$$\frac{\partial \phi}{\partial t} + \frac{p}{\rho} + \frac{1}{2} |\nabla \phi|^2 + \Upsilon = C(t)$$  \hspace{1cm} (2.12)

where $C(t)$ is a function of time only and can be determined from initial and boundary conditions.

Having derived the governing equations for unsteady potential flow, it is necessary to address the general approach implemented to find a solution and to extract useful information from that solution. Often in potential flow based aerodynamics, the lifting surfaces and wakes are modeled by superimposing solutions to the Laplace equation (Eq. 2.10), referred to as singularity elements. The strengths of these singularity elements are determined through the application of boundary conditions. This process ultimately leads to a system of linear equations, which can be solved computationally using standard numerical methods. Once the strength of all singularity elements are known, the velocity throughout the flowfield is defined. With this information, the inviscid forces on the lifting surfaces, which are physically a result of the pressure differential across the surface, can be found with the unsteady Bernoulli equation (Eq. 2.12).
2.3.2 The Distributed Vorticity Element

The singularity element used in this method is the distributed vorticity element (DVE) of Bramesfeld [8] and Bramesfeld and Maughmer [45], which consists of a combination of two discrete vortex filaments, with circulations that vary parabolically along the span, and two semi-infinite sheets of vorticity. The strength of each sheet must vary linearly such that in combination with its respective discrete vortex filament, Helmholtz's theorems regarding vorticity are satisfied. This combination of a discrete vortex filament and a semi-infinite sheet of vorticity is a second-order lifting line. Thus, the discrete vorticity element is the coplanar superposition of two of these lifting lines of opposite orientation, as shown in Fig. 2.1.

As indicated in Fig. 2.1, the form of the circulation distribution in units of length squared per time (e.g. $m^2/s$) over the discrete vortex filament is

$$\Gamma(\eta) = A + B\eta + C\eta^2$$  \hspace{1cm} (2.13)

where $\eta$ is the spanwise coordinate in the local coordinate system in units of length.
This coordinate system is centered at the mid-span and mid-chord location of the $i^{th}$ DVE, such that this DVE spans from $-\eta_i$ to $+\eta_i$. The chordwise local coordinate is $\xi$. The $\zeta$ axis is defined orthogonal to the $\xi$ and $\eta$ axis. The control point of the DVE is the origin of the local coordinate system.

The leading and trailing edge filaments of a single DVE have opposite orientations, so if the leading-edge coefficients are positive, the trailing-edge coefficients are negative and vice versa. The strength of the vortex sheet is the derivative of the circulation distribution, viz.

\[
\gamma(\eta) = \frac{\partial \Gamma(\eta)}{\partial \eta} = B + 2C\eta
\]  

(2.14)

where the units of $(\gamma)$ are length over time (e.g. $m/s$).

If the bound vortex of the lifting line is unswept relative to the sheet, as shown in Fig. 2.2, the potential function can be written in a closed form as determined originally by Von Karman [65] and later simplified by Wu [66]. This closed form is given by

\[
\phi(\xi_0, \eta_0, \zeta_0) = \frac{\zeta_0}{4\pi} \int_{-\eta_i}^{\eta_i} \frac{\Gamma(\eta)}{(\eta_0 - \eta)^2 + \zeta_0^2} \left(1 + \frac{\xi_0}{\sqrt{\zeta_0^2 + (\eta_0 - \eta)^2 + \zeta_0^2}}\right) d\eta 
\]  

(2.15)

where $\xi_0$, $\eta_0$, and $\zeta_0$ are coordinates in a local reference frame centered at the mid-span location of the lifting line, and the span of the lifting line is $2\eta_i$. The addition of a second lifting line of opposite orientation and two simple coordinate transformations to the center of the DVE, as defined in Fig. 2.1, results in the
potential function

\[
\phi(\xi_0, \eta_0, \zeta_0) = \frac{\zeta_0}{4\pi} \int_{-\eta_i}^{\eta_i} \left[ \frac{\Gamma(\eta)}{(\eta_0 - \eta)^2 + \zeta_0^2} \left( 1 + \frac{\xi_0 + \xi_i}{\sqrt{(\xi_0 + \xi_i)^2 + (\eta_0 - \eta)^2 + \zeta_0^2}} \right) + \frac{-\Gamma(\eta)}{(\eta_0 - \eta)^2 + \zeta_0^2} \left( 1 + \frac{\eta_0 - \xi_i}{\sqrt{(\eta_0 - \xi_i)^2 + (\eta_0 - \eta)^2 + \zeta_0^2}} \right) \right] d\eta
\]

(2.16)

Figure 2.2: Simple lifting line whose potential function is given by Eq. 2.15.

This solution is limited in that it does not account for leading or trailing edge sweep. A simple closed form solution for the potential function of such an element has yet to be developed. It is, however, possible to build a closed-form, composite solution in the velocity domain for an element having a swept leading or trailing edge using the three-dimensional Biot-Savart law as applied to each of the four fundamental solutions (leading-edge filament, leading-edge sheet, trailing-edge filament, and trailing-edge sheet). Because both the boundary conditions and resulting forces depend on velocity and not directly on the value of the potential
function, and because the velocity is a linear function of the potential and thus also obeys the superposition principle, it is possible to forgo the potential function entirely in favor of the velocity distribution. Thus, as derived by Bramesfeld [8], the velocity induced by a DVE at a point in the flow field in the local coordinate system defined in Fig. 2.1 is

\[
V_i(\xi_0, \eta_0, \zeta_0) = V_{i,\text{filament}}(\xi_0 + \xi_{LE}, \eta_0, \zeta_0, A, B, C, \varphi_{LE}) + \\
V_{i,\text{filament}}(\xi_0 + \xi_{TE}, \eta_0, \zeta_0, -A, -B, -C, \varphi_{TE}) + \\
V_{i,\text{sheet}}(\xi_0 + \xi_{LE}, \eta_0, B, C, \varphi_{LE}) + \\
V_{i,\text{sheet}}(\xi_0 + \xi_{TE}, \eta_0, -B, -C, \varphi_{TE})
\]  

(2.17)

The expansion of the terms included in Eq. 2.17 can be found in Appendix A.

### 2.4 Modeling of the Lifting Surfaces and Wakes

#### 2.4.1 Lifting Surfaces

The body-fixed reference frame, which is coincident with the global reference frame at the start of computations, is defined within the HOFW method for conventional fixed-wing aircraft and shown in Fig. 2.3. The x-z plane is defined as the symmetry plane for conventional aircraft, such that the x-y plane is that of the projected wing planform with the positive y-axis is defined out the right wing. Positive twist (ε) and angel of attack (α) of the wing are defined according to the right-hand rule about the y-axis.
Figure 2.3: The body-fixed coordinate system.

The DVEs used to parametrize the lifting surface are referred to as surface DVEs (SDVEs), and represent the geometry of the lifting surface as shown for a notional single-lifting line example in Fig. 2.4. Regardless of the number of lifting lines, the total chord length at the edge of each SDVE for each section is equivalent to the chord length of the wing at that location. For a single row of SDVEs, the leading-edge vortex filament of the SDVE is placed along the quarter-chord line of the wing. This placement is based on the thin-airfoil-theory result that the center of pressure (and thus the location at which the lift acts) is at the quarter chord of the wing. To reproduce the lift-curve slope according to thin-airfoil theory, the control point must then be placed at the three-quarter chord of the wing. Because the control point of the SDVE is located at its mid-span and mid-chord location, placement at the three-quarter chord of the wing necessitates that the trailing-edge vortex filament extend beyond the trailing edge of the wing by one-quarter chord length. This geometric representation, as it is based on thin airfoil theory, accounts for smooth flow off of the trailing edge of the wing (the Kutta condition) [44].

As the number of lifting lines is increased, the trailing edge remains at three-
quarters of the length of the aft-most row of SDVEs. Because the total chord length of the SDVE model is a constant (equivalent to the chord length of the wing), this necessitates that the leading edge move closer to the leading edge of the wing. In the limit, as the number of lifting lines increases, the SDVE geometry approaches the exact wing geometry. Once again, the overhang of the most aft row of SDVEs at the trailing edge allows for a better representation of the Kutta condition.

![Diagram of surface DVEs](image)

Figure 2.4: An example of the conversion of a simple planar wing geometry to one chordwise row of surface DVEs.

Application of the Kutta condition in unsteady cases is valid over a limited range of conditions as well, according to Katz and Plotkin [44]. The first limitation is that trailing-edge separation must not be present, appropriate only for low to moderate angles of attack. In the same vein, to prevent trailing-edge separation with increasing reduced frequencies \( \frac{\omega c}{2V_\infty} \), the trailing-edge displacement amplitude must decrease. Finally, it is recommend that a limitation be placed on the vertical kinematic velocity of the trailing edge \( \dot{h} \) in all cases, such as airfoil
pitching or plunging, such that $\frac{h}{v_\infty} << 1$ [44].

### 2.4.2 Rotating Lifting Surfaces

To model both the wing and propeller, the propeller geometry is defined with respect to the wing coordinate system. The primary challenge in this conversion is correctly accounting for the pitch angle of the propeller blade ($\beta$) in terms of the incidence angle ($\epsilon$). The pitch angle is defined relative to the plane of rotation, while incidence angle is defined about the y-axis of the body-fixed, fixed-wing reference frame (positive out the right wing). If the plane of rotation is assumed to lie in the y-z plane, an incidence angle of $90^\circ$ or $-90^\circ$ corresponds with a pitch angle of zero degrees. An example containing only positive rotations is provided in Fig. 2.5 to aid in visualization of this issue. Because the leading edge location is defined, the selection of $90^\circ$ or $-90^\circ$ rotation depends on the direction of rotation. If the rotation is in the positive (according to the right-hand rule) x-direction as shown in Fig. 2.5, the incidence angle is given by $\epsilon = 90 - \beta$. If the rotation is in the negative x-direction, the incidence angle is given by $\epsilon = -90 + \beta$.

Once the first blade is fully defined at the zero azimuthal location, the geometry is repeated and rotated to define the remaining propeller blades. When the propeller is fully defined, the entire geometry is translated to the defined hub location for that propeller in three-dimensions within the global coordinate system. This process is repeated for the total number of propellers specified.

### 2.4.3 Compressibility Corrections

Small amounts of compressibility can be accounted for through application of the Prandtl-Glauert similarity rule [44]. This is a simple correction in which the ge-
Figure 2.5: The propeller pitch angle \( \beta \) is related to the incidence angle \( \epsilon \) via a 90\(^\circ\) rotation, as shown for a simple rectangular propeller blade rotating in the positive direction about the x-axis.

Geometry of each lifting surface is modified such the coordinate in the free-stream direction is scaled by a ratio \( \beta \), where

\[
\beta = \frac{1}{\sqrt{1 - M_{\infty}^2}} \quad \text{(2.18)}
\]

For example, if the free-stream velocity is parallel to the x-axis, modified coordinates of the wing would be given by

\[
\begin{align*}
    x_{\text{mod}} &= \beta x \\
    y_{\text{mod}} &= y \\
    z_{\text{mod}} &= z
\end{align*} \quad \text{(2.19)}
\]

The identification of the free-stream direction for a propeller-wing system is complicated by interactions between the surfaces. Assuming that interaction effects are small relative to the true free-stream velocity on the wing and the sum of the
free-stream and rotational velocities on the propeller, the interaction effects may be neglected for this application. This correction can thus be applied prior to any calculations through adjustment of the wing and propeller geometries based on known operating conditions.

### 2.4.4 Wake Development

The wake of each lifting surface is developed using time-stepping within the HOFW method, both to permit time-dependent analysis and because a time-stepping approach allows for better computational efficiency than an equivalent spacial relaxation method [44]. In this approach, each SDVE is advanced at a distance that depends on the velocity of the lifting surface in the global reference frame and the predefined time-step size. A wake DVE is formed in the space spanning the previous time-step SDVE trailing-edge location and the current time-step SDVE trailing-edge location. In relaxed-wake analysis, this shed wake element is then permitted to deform according to the velocities induced by the full system prior to the next time-step.

The form of a wake DVE depends on whether the analysis is steady or unsteady. A depiction of the difference between these two modes is provided in Fig. 2.6. If the flow field is taken to be steady, the strengths of the vortex sheets in the wake are based on the circulation of the trailing-edge SDVEs and, because they cancel, the leading and trailing-edge vortex filaments can be omitted from the calculation of velocities modified by the wake. The wake developed with this process is a continuous vortex sheet and as such induces finite velocities everywhere except for logarithmic singularities at the outer most edges of the sheet [8, 45]. If the flow field is unsteady, the leading and trailing edge filaments of the wake DVEs do not
necessarily cancel and must be retained. In this case, the presence of singular vortex filaments within the wake may influence the stability of the solution.

![Image of Quasi-steady Mode and Unsteady Mode](image.png)

**Figure 2.6:** A comparison of unsteady versus steady wake development as shown notionally with a fixed-wake.

The treatment of the wake DVEs during each time step also depends on the mode of analysis. During both fixed- and relaxed-wake analysis, the wake DVEs are initially shed parallel to the kinematic velocity of the lifting surface. The term “kinematic velocity” is used here rather than “free-stream velocity” because the kinematic velocity of a rotating lifting surface includes the velocity resulting from rotation of the surface in addition to the free-stream velocity. During a fixed-wake analysis, the wake DVEs are of constant size, shape, and location in the global coordinate frame and thus remain in their initial location indefinitely. During a relaxed-wake analysis, each element is permitted to translate and stretch according to the local velocities as calculated at the mid-chord points at the side edges.

The advection of the DVEs with the local velocity is defensible in both the steady and unsteady case due to Kelvin’s theorem of vorticity [67], which states
that for a specified set of fluid particles that define a closed contour in an inviscid, barotropic flow with conservative body forces,

\[
\frac{D\Gamma}{Dt} = 0
\]  

(2.20)

If a closed contour is assumed to exist on the upper surface of the DVE (parallel to the plane of the DVE), the net circulation (\(\Gamma\)) of that contour is zero (as it would contain no vorticity). A contour containing the same fluid particles after some time has passed will also have zero circulation. This would also be true for a contour placed on the lower surface of the DVE, or in front of, behind, or beside the DVE. Thus, the DVE is in effect trapped within the local flow field and must move accordingly.

Another result of Eq. 2.20 is that the strength of the vortex sheet of the DVE must adjust as it deforms with the local velocity such that the integrated circulation is constant with time. This can be demonstrated simply by placing the closed contour such that it contains the vorticity of the DVE. Because the circulation enclosed by the contour cannot change with time, the strength of the vortex sheet must adjust to maintain the initial integrated circulation within the spatially adjusted contour.

While this process does create a force-free wake in both steady and unsteady flows, it does not necessarily force the vortex sheet to be aligned with the local velocity. Rather, it indicates that the perpendicular component of the local velocity relative to the vorticity must be zero, i.e. \(V_r \times \gamma = 0\). This can be accomplished either because the vortex sheet is aligned with the local velocity (in which case both terms may be non-zero but their cross product is still zero), or because the vortex sheet is advected with the local velocity (in which case the relative velocity
is zero regardless of the orientation of the DVE).

The former case is necessarily true for steady flows, as the vortex lines must be aligned with the streamlines in order for Eq. 2.20 to be true. This is illustrated in Fig. 2.7, where a group of fluid particles move within a steady flow field from the region bounded by the contour $s$ at an instant in time to the region bounded by the contour $s'$ at a later instant in time along a streamline. In order for the integrated circulation along the contour to be constant for $s$ and for all other possible contours within the flow field, a streamline cannot cross the vortex line.

Figure 2.7: Kelvin’s theorem of vorticity as applied to a steady flow within the context of an aircraft wing.

In the case of unsteady flows, however, the streamlines and vortex lines are not necessarily coincident. The streamlines in an unsteady flow are a function of time and are not equivalent to the pathlines that define the motion of the fluid elements. This difference means that a force-free wake can exist in which the vortex lines are not aligned with the streamlines at an instant in time. For example, a two-dimensional potential vortex could be perpendicular to a pathline and still satisfy Eq. 2.20 as long as it advects along the pathline resulting in a zero relative
velocity. This case is illustrated in Fig. 2.8.

Figure 2.8: Kelvin’s theorem of vorticity as applied to an unsteady flow in the context of an aircraft wing.

In modeling a steady case, the wake DVEs include only the vortex sheets (no leading and trailing edge vortices), and these sheets are adjusted instantly according to the circulation shed from the upstream surface DVE. The solution is initially time-dependent, but as the wake DVEs are advected along local velocities, they form a wake that changes relatively little in shape in the near field. In this case, the advection results in convergence to a steady condition in which the vorticity is aligned with the streamlines/pathlines, such as is illustrated in Fig. 2.7. In mod-
eling an unsteady case, the wake DVEs include their leading and trailing vortices. The advection of the wake DVEs with the local velocities in this case will include the bulk motion of shed vorticity (at each intersection between wake elements), such as that illustrated in Fig. 2.8.

### 2.4.5 Special Kinematic Considerations for Rotating Surfaces

The motion of a propeller SDVE consists of both rotation and translation during each time-step. To complete this process, three points on each DVE are translated via a kinematic velocity: the right end of the leading edge of the DVE, the left end of the leading edge of the DVE, and the mid-chord mid-span location. The cross product of the vector between the leading edge endpoints and the vector between the midpoint along the leading edge and midpoint mid-chord location defines the normal vector for the SDVE in its new location and, with this information, the orientation and geometry of the SDVE at its new location are completely defined.

The velocity of a location on a propeller blade at an instant in time is the vector sum of the velocity induced by rotation and the free-stream velocity, viz.

\[
\vec{V}_{i,r} = \vec{\omega} \times \vec{r}_i + \vec{V}_\infty
\]

The rotational velocity is dictated within the computational framework for each propeller via an individually defined advance ratio. This accounts for both propellers operating at different rotational rates and for oppositely oriented rotation via a negative advance ratio. In the latter case, the geometry of the propeller is adjusted appropriately for the direction of rotation as explained in Section 2.4.2. The free-stream velocity is dictated for the full propeller-wing system through the
motion of the system (permissible due to the Galilean invariance of the governing equations) and separated into global x-, y-, and z-components via a prescribed angle of attack and side-slip angle.

It is necessary to delineate between the motion of each SDVE as it pertains to its location within the global reference frame (kinematic velocity) and the time-accurate local fluid velocity experienced by each SDVE as is used to determine circulation and surface forces. In fixed-wing analysis, this delineation is unnecessary, as the velocity of each SDVE within the global reference frame is equal and opposite to the defined free-stream velocity. If the total velocity, \( V_{i,r} \), is used to move the propeller blade in the model, the blade moves progressively outward from the axis of rotation, as shown in Fig. 2.9, in which the \( i^{th} \) DVE moves from position one to position two. This is an intuitive result, as the velocity of an SDVE at any instant in time is perpendicular to the radius from the axis of rotation. To discretize the motion of the propeller blade, the velocity of each SDVE must be calculated in such a way as to move the DVE from position one to position three. Mathematically, this velocity is calculated as

\[
V_{i,d} = \frac{\mathbf{P}_{i,(1-3)}}{\Delta t}
\]

(2.22)

where \( \mathbf{P}_{i,1-3} \) is the vector from the \( i^{th} \) DVE’s current location to its location after the next time step, and \( \Delta t \) is the time-step size.

The use of \( V_{i,d} \) as the local fluid velocity results in significant discretization errors, both in terms of the calculated circulation and the SDVE forces. To address this error, \( V_{i,r} \) is inserted into the resultant matrix and used as the local fluid velocity for force calculations. The difference between using the discretized velocities and the ideal velocities for this purpose is highlighted by considering
Figure 2.9: The velocity designated for each SDVE on a propeller blade must be defined such that the SDVE arrives at the correct location in space. This requires a delineation between the rotational velocity used for aerodynamic analysis ($V_{i,r}$) and that used to define the motion of the blade ($V_{i,d}$).

The difference in time-step size required for convergence of the integrated thrust and torque. For a representative case with discretized velocities, a time-step size resulting in over 100 time steps per revolution was required to converge in terms of calculated thrust within 1%. For the same case with ideal velocities, convergence is achieved between 10 and 20 time steps per revolution.

### 2.5 Numerical Solution Method

The application of the HOFW method to any design problem requires a computational framework to define the geometry and motion of the lifting surfaces such that design parameters can be varied without expending significant time and effort. Having to manually calculate and implement the geometry, location, and velocity of every SDVE in a design at each time-step, for example, would nullify any time savings achieved by solving the governing equations quickly. Likewise, implementation of the solution method itself would be very tedious by hand. These
problems were addressed by previous studies for fixed-wing systems, and the reader is referred both to the previous publications on the HOFW method [8, 45] and to similar methods described in Katz and Plotkin [44] for further reading.

A flow chart detailing the overall computational approach is shown in Fig. 2.10. The steps shown are divided into two main components: initialization and time-step loop. During the initialization, the geometry and operating conditions are prescribed in the global reference frame, thereby defining the body-fixed and DVE reference frames. From this location, the time-step loop is entered, during which the propeller-wing system is moved in the global coordinate frame according to the defined operating conditions.
Figure 2.10: A flow chart of the HOFW method as modified for analysis of propeller-wing systems.

The bulk of the solution process involves the determination of the circulation distribution over each SDVE at each time step, which can be formulated as
\[ [A] \{x\} = \{B\} \] (2.23)

where \([A]\) is the influence matrix, which is a function of the geometry of the set of surface DVEs, \(\{x\}\) is the vector containing the unknown strengths of the surface DVEs, and \(\{B\}\) is the resultant vector in which the boundary conditions are applied. The strength of each surface DVE is defined by the coefficients to the circulation distribution \((A, B, \text{and } C)\) as given in Eq. 2.13. Thus, there are three unknowns for each DVE, so the \(\{x\}\) vector for a system with \(N\) DVEs is \(3N\) elements long.

To solve for the three coefficients on each DVE, three boundary conditions are required. The first boundary condition is that the flow must be tangent to the surface at each control point (equivalent to a von Nuemann boundary condition). The equation provided by this boundary condition is of the form

\[ V_\infty \cdot \bar{n}_i + \bar{w}_i \cdot \bar{n}_i = 0 \] (2.24)

where \(i\) indicates the \(i^{th}\) DVE, \(n\) is the normal vector, and \(w\) is the velocity induced by the sum of all other surface and wake DVEs. The wake DVE induced velocities are based on the wake strength and geometry as determined in the previous time-step. Thus, grouping known values on the RHS, Eq. 2.24 becomes

\[ \nabla_{i,\text{surface}} \cdot \bar{n}_i = -V_\infty \cdot \bar{n}_i - \nabla_{i,\text{wake}} \cdot \bar{n}_i \] (2.25)

where the left hand side is a function of the geometry of the lifting surface and the unknown circulation coefficients and the right hand side is a function of the known wake shape and solution at the previous time-step. Thus, Eq. 2.25 represents \(N\)
equations with $3N$ unknowns.

Of the remaining boundary conditions, $2N - 2$ come from enforcing that both
the circulation distribution and vortex-sheet strength must be continuous on the
lifting line. Specifically, continuity of the circulation distribution results in $N - 1$
equations of the form

$$A_i + B_i \eta_i + C_i \eta_i^2 - A_{i+1} + B_{i+1} \eta_{i+1} - C_{i+1} \eta_{i+1}^2 = 0$$  \hspace{1cm} (2.26)$$

Likewise, $N - 1$ additional equations are provided through enforcement of contin-
nuity of the vortex sheet strength, as in

$$B_i + 2C_i \eta_i - B_{i+1} + 2C_{i+1} \eta_{i+1} = 0$$  \hspace{1cm} (2.27)$$

The two remaining boundary conditions come from forcing the circulation at each
free tip of a lifting surface to zero, i.e.

$$A_1 + B_1 \eta_1 + C_1 \eta_1^2 = 0$$
$$A_N + B_N \eta_N + C_N \eta_N^2 = 0$$  \hspace{1cm} (2.28)$$

As an example of this process, consider the simple DVE system shown in Fig.
2.11. The first row of DVEs (labeled 1–3) are surface DVEs. The second two rows
of DVEs are wake DVEs (labeled 4–9). Each DVE has its own local coordinate
system ($\eta, \xi, \zeta$) centered at the element centroid, which is also the location of the
control point on the surface DVEs. To simplify the mathematics, the DVEs in this
example can be assumed to be planar and at a small angle of attack relative to
the free-stream velocity, which can be assumed to be in the global x-direction.
Each SDVE has three unknowns, and thus the \( \{ x \} \) vector from Eq. 2.23 for the system shown in Fig. 2.11 is

\[
\{ x \} = \begin{bmatrix}
A_1 \\
B_1 \\
C_1 \\
A_2 \\
B_2 \\
C_2 \\
A_3 \\
B_3 \\
C_3
\end{bmatrix}
\] (2.29)

The influence matrix, \( [A] \) from Eq. 2.23, holds the coefficients to the left hand terms in Eqs. 2.25, 2.26, 2.27, and 2.28, while the resultant vector \( \{ B \} \) from Eq. 2.23 holds the right hand terms. Eqs. 2.26, 2.27, and 2.28 thus provide the first six rows in the influence matrix and resultant vector, and are straightforward to
implement. Moving left to right, the boundary conditions are as follows: Zero circulation at left end of SDVE 1,

\[ A_1 + B_1(-\eta_1) + C_1(-\eta_1)^2 = 0 \]  \hspace{1cm} (2.30)

continuity of circulation between SDVE 1 and 2 and SDVE 2 and 3,

\[ A_1 + B_1\eta_1 + C_1\eta_1^2 - A_2 + B_2\eta_2 - C_2\eta_2^2 = 0 \]  \hspace{1cm} (2.31)

\[ A_2 + B_2\eta_2 + C_2\eta_2^2 - A_3 + B_3\eta_3 - C_3\eta_3^2 = 0 \]  \hspace{1cm} (2.32)

continuity of vortex-sheet strength between SDVE 1 and 2 and SDVE 2 and 3,

\[ B_1 + 2C_1\eta_1 - B_2 + 2C_2\eta_2 = 0 \]  \hspace{1cm} (2.33)

\[ B_2 + 2C_2\eta_2 - B_3 + 2C_3\eta_3 = 0 \]  \hspace{1cm} (2.34)

and finally, zero circulation at the right end,

\[ A_3 + B_3(\eta_3) + C_3(\eta_3)^2 = 0 \]  \hspace{1cm} (2.35)

The last three rows of the influence matrix are a result of Eq. 2.25 as applied at the three control points on DVEs 1 – 3. As an example, Eq. 2.25 applied for DVE 1 results in

\[ \nabla_{i,1,surface} \cdot \vec{n}_1 = -\nabla_{\infty} \cdot \vec{n}_1 - \nabla_{i,1,wake} \cdot \vec{n}_1 \]  \hspace{1cm} (2.36)
The induced velocities, $V_{i,\text{surface}}$ and $V_{i,\text{wake}}$, must be expanded based on the equations found in Appendix A. The left hand term, $V_{i,\text{surface}}$, is the sum of the velocities induced by the SDVEs at the control point on DVE 1, as in

$$V_{i,\text{surface}} = V_{i,DVE_1}(x_1, y_1, z_1) + V_{i,DVE_2}(x_1, y_1, z_1) + V_{i,DVE_3}(x_1, y_1, z_1)$$  \hspace{1cm} (2.37)$$

The induced velocities of each SDVE are given by the equations found in Appendix A, and are a linear function of the circulation coefficients of that SDVE and the location of control point 1 in each SDVE’s local coordinate frame. Using the functions defined in Appendix A, the influence matrix for this system is thus completed as

$$[A] = \begin{bmatrix}
1 & \eta_1 & \eta_1^2 & 0 & 0 & 0 & 0 & 0 & 0 \\
1 & \eta_3 & \eta_3^2 & -1 & -\eta_2 & \eta_2^2 & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & \eta_2 & \eta_2^2 & -1 & \eta_3 & -\eta_3^2 \\
0 & 1 & 2\eta_1 & 0 & -1 & 2\eta_2 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 1 & 2\eta_2 & 0 & -1 & 2\eta_3 \\
0 & 0 & 0 & 0 & 0 & 1 & \eta_3 & \eta_3^2 & \eta_3^2 \\
\vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\
\vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\
\end{bmatrix}$$ \hspace{1cm} (2.38)$$

where

$$H_1 = \begin{bmatrix}
(a_{1\xi,\text{LE}} - a_{1\xi,\text{TE}})\hat{x} \\
(a_{1\eta,\text{LE}} - a_{1\eta,\text{TE}})\hat{y} \\
(a_{1\zeta,\text{LE}} - a_{1\zeta,\text{TE}})\hat{z}
\end{bmatrix} \cdot \hat{n}_1$$ \hspace{1cm} (2.39)$$
\[
H_2 = \begin{bmatrix}
(b_{1\xi,LE} - b_{1\xi,TE})\hat{x} \\
(b_{1\eta,LE} - b_{1\eta,TE} + b_{2\eta,LE} - b_{2\eta,TE})\hat{y} \\
(b_{1\zeta,LE} - b_{1\zeta,TE} + b_{2\zeta,LE} - b_{2\zeta,TE})\hat{z}
\end{bmatrix} \cdot \hat{n}_1
\]

(2.40)

and

\[
H_3 = \begin{bmatrix}
(c_{1\xi,LE} - c_{1\xi,TE})\hat{x} \\
(c_{1\eta,LE} - c_{1\eta,TE} + c_{2\eta,LE} - c_{2\eta,TE})\hat{y} \\
(c_{1\zeta,LE} - c_{1\zeta,TE} + c_{2\zeta,LE} - c_{2\zeta,TE})\hat{z}
\end{bmatrix} \cdot \hat{n}_1
\]

(2.41)

Although not visible in this formulation, \(H_1\) through \(H_3\) depend on the location of the control point, so the last three rows in the influence matrix are independent.

For simplicity, in this system the local coordinate systems are aligned with the global coordinate system, eliminating the need for coordinate system transformations. In most practical applications, this is not the case and both the induced velocities and normal vector must be transformed into the global coordinate frame.

The resultant vector, \(R\), consists of the right hand side of Eqs. 2.30 through 2.36. Thus, the upper two-thirds of the resultant matrix, corresponding to the continuity conditions, are zeros. The lower third of the resultant matrix is the component of the free-stream and wake induced velocities normal to each surface DVE at their respective control points. This again requires the use of the equations found in Appendix A, but in this case, the coefficients for each wake DVE are known apriori, either because they were updated in the previous time-step to the shed vorticity (as in the steady case) or because they are constant with time (as in the unsteady case). For example, the term in the resultant vector corresponding to the flow tangency boundary condition at the control point on DVE 1 is
\[-\nabla_\infty \cdot \vec{n}_1 - \nabla_{i,1,\text{wake}} \cdot \vec{n}_1 = -\nabla_\infty \cdot \vec{n}_1 -
\]
\[
\begin{bmatrix}
\nabla_{i,DVE4}(x_1, y_1, z_1) - \nabla_{i,DVE5}(x_1, y_1, z_1) - \\
\nabla_{i,DVE6}(x_1, y_1, z_1) - \nabla_{i,DVE7}(x_1, y_1, z_1) - \\
\nabla_{i,DVE8}(x_1, y_1, z_1) - \nabla_{i,DVE9}(x_1, y_1, z_1)
\end{bmatrix} \cdot \vec{n}_1
\]

(2.42)

Thus, the resultant vector is

\[
R = \begin{bmatrix}
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
-\nabla_\infty \cdot \vec{n}_1 - \nabla_{i,1,\text{wake}} \cdot \vec{n}_1 \\
-\nabla_\infty \cdot \vec{n}_2 - \nabla_{i,2,\text{wake}} \cdot \vec{n}_2 \\
-\nabla_\infty \cdot \vec{n}_3 - \nabla_{i,3,\text{wake}} \cdot \vec{n}_3
\end{bmatrix}
\]

(2.43)

and this system of equations is solved numerically using Gaussian elimination.

The next steps within the time-step loop depend on the mode of analysis. If the analysis mode is steady, the wake strength is updated according to the new circulation distribution due to the assumption that under steady conditions, the influence of a change in vorticity on the surface is instantaneously propagated throughout the wake. If the analysis is time-dependent, the new circulation distribution will be shed into the next row of wake DVEs at the beginning of the next time-step to account for shed circulation effects. The development of the fixed or relaxed wake
then takes place as described in Section 2.4.

### 2.6 Clarification of Modes of Analysis for Propeller-Wing Systems

To aid in understanding of the results and the approach to analysis, a flowchart detailing the potential modes of analysis is provided in Fig. 2.12. Overarching all other considerations is whether or not to include profile drag. This decision is entirely up to the user and is not influenced by any other conditions. The default (indicated in green in the flowchart) is that profile drag is included.

First, it is prudent to discuss the treatment of the wake. If the wake is “fixed,” this selection applies to all lifting surfaces (both the propeller and wake). Likewise, if the wake is treated as “relaxed,” this is a universal selection and all wake DVEs are assumed to convect with the local velocity. The inclusion of unsteady effects in terms of the wake requires the inclusion of shed circulation in the wake. In contrast to the relaxation parameter, the unsteady treatment applies only to the wakes of fixed (non-rotating) lifting surfaces. This is primarily to prevent the shed circulation that would be present in the propeller wake due to asymmetric inflow conditions from adversely effecting the solution on the downstream wing surface.

In terms of the calculation of forces, the unsteady treatment includes accounting for both shed vorticity and non-circulatory terms. These again, apply only to fixed surfaces in this formulation. If the system is not treated as unsteady, it is assumed to be “quasi-steady,” which is to say that the solution is still time-dependent in that flow tangency is enforced at every time-step, but that changes in circulation on the lifting surfaces are instantly propagated through the wake. In this case,
Figure 2.12: Flowchart detailing the possible modes of analysis within the HOFW method. Overall default options are indicated in green. The default options for unsteady analysis are outlined in purple.
both shed circulation and non-circulatory effects are neglected.

Although it is rigorously defensible to use a relaxed-wake with the inclusion of unsteady effects (as discussed in the previous sections), in practice this combination was often found to lead to instability within the wake and non-realistic solutions due to the singular leading-edge and trailing-edge vortices. As a result, all fully unsteady results provided in this thesis were calculated with a fixed, drag-free wake. Further addressing of the influence of rotating surface lift unsteadiness on the performance of the rotating lifting surface and propeller-wing interaction are left as future work.

The majority of analysis completed within this thesis were taken as quasi-steady with a relaxed wake and inclusion of profile drag effects. Any deviation from this mode of analysis, as in Chapter 5 (which focuses on unsteady effects) is explicitly stated.

2.7 Framework for Rapid Design-Space Exploration

One aspect of the current approach which provides particular utility for design-space exploration is its simplified approach to geometric definition of lifting surfaces. The lifting surfaces (as described in Section 2.4) are defined with the quarter-chord location, twist, and chord of each panel endpoint within the body-fixed coordinate system. Only a single propeller blade need be defined within the input file. This propeller blade is then repeated, rotated, and translated according to the input number of blades per propeller, number of propellers, propeller axis location, and propeller rotation direction (as dictated by the individual advance
ratio). Finally, the airfoil for each panel is defined by the specification of a look-up file, which spans a range of Reynolds numbers. This simplified approach to lifting surface definition, combined with a relatively low runtime and ability to evaluate complex interactions, provides a method well suited for design-space exploration of complex propeller-wing systems.

The simplicity of the geometric input process as compared to that required for CFD is noteworthy. In a CFD method, the definition and detailed mesh generation process for a similar design-space study is significantly more complex. For example, an investigation of the spanwise chord distribution on a wing within a propeller-wing system using the HOFW method requires adjustments to the chord length input only. The airfoil Reynolds number range will account for any changes in local Reynolds number, and because thickness is not elsewhere accounted for, there is no need to adjust the panel resolution. Similarly, a change in rotation direction requires only a change in sign of the advance ratio. In contrast, each design change in CFD would require a redefinition of the geometry details, i.e., cross section and variations between cross sections. In addition, each configuration would require re-meshing and a re-examination of each mesh to ensure a “valid, high quality mesh”, both of which currently require user attention [55].

The ease of geometric definition in combination with the reduced computational resources required for finding a solution with the HOFW method enables rapid design-space exploration that challenges the current state of the art of CFD.

2.8 Summary

Within this chapter, the foundations of the HOFW method have been described. The flowfield solution requires inviscid, irrotational, and incompressible flow. Some
viscous and compressibility effects are accounted for through the addition of profile drag (as will be explained more thoroughly in the proceeding chapter) and compressibility can be accounted for additionally through adjustment of the lifting-surface geometry as determined by the Prandtl-Glauert similarity rule. Finally, the approach to solving for the strength of the DVEs to define the flowfield has been explained and a simple example has been provided. This is accomplished through construction of a system of linear equations based on the boundary conditions for each SDVE, including continuous circulation and vortex-sheet strength between adjacent SDVEs, zero circulation at the free edges of each wing, and flow tangency at the control point. The system of equations is then solved via Gaussian elimination. The approach allows for rapid design space exploration due to the ease of geometric definition and low computational-resource requirements for finding a numerical solution.
Chapter 3

Calculation of Forces and Limitations of Method

3.1 Overview

In the previous chapter, the approach that is used to determine the strength and location of all singularity elements within the flow-field was addressed. The objective of this chapter is to describe the approach to the calculation of lift and drag forces with specific regard to the applicability of the approach within an unsteady flow domain. In addition, limitations of the HOFW method are identified and addressed as they apply to propeller-wing systems.

3.2 Lift

3.2.1 Discussion of Unsteady Lift

Once the full flow solution is determined, the lift and drag force are calculated for each SDVE. On a real wing, these forces are physically the result of the integrated
pressure over the surface of the wing resolved in the lift and drag directions (per-
pendicular and parallel to the free-stream velocity, respectively). Because the wing
is approximated as a thin lifting surface within the HOFW method, the physical
surface is not represented. As a result, a true pressure integration is not possible.
A pressure integration over the thin lifting surface approximation is possible but
is inherently lacking in resolution, particularly in the region of the leading edge.
To account for the true leading-edge pressure distribution, the thin lifting surface
representation must be augmented with a leading-edge suction model. An alter-
native technique that implicitly accounts for the leading-edge suction force in the
steady limit is the unsteady Kutta-Joukowski approach.

Although the unsteady Kutta-Joukowski approach has been presented and used
previously [68], a rigorous proof of the method is not available in the literature. To
support the method, the unsteady lift as calculated through pressure integration
on a two-dimensional flat plate is described in this section. Full details of the
derivation of this calculation are provided in Appendix B. The description in this
section provides an overview of the derivation and a direct link between the results
of such an approach and the terms included in the unsteady Kutta-Joukowski equa-
tion, offering insight into both the physical source of the terms and the differences
between the approaches.

The discussion that follows considers a two-dimensional strip of a lifting sur-
face. Application of the approach therefore assumes high aspect ratios with little
sweep, as pressure differences due to spanwise flow are neglected. To be consistent
with the method itself, the derivation uses three coordinate systems: the global
coordinate system (x, y, and z positions and u, v, and w velocities), the aerody-
namic coordinate system based on the free-stream velocity (lift, side-force, and
drag), and the local surface coordinate system \((\xi, \eta, \zeta)\). These three coordinate systems are shown with respect to a section of a lifting surface in Fig. 3.1. In the derivation provided in Appendix B, the strip of the lifting surface begins as a continuous distribution of vorticity in the chordwise direction. In addition, to simplify the mathematics, the three coordinate systems are assumed to have coinciding origins at the spanwise center of the leading edge of the lifting surface, and the \(y\)-, side-force, and \(\eta\) axes are aligned.

![Figure 3.1: Angle and coordinate system definitions for pressure integration on a two-dimensional flat plate.](image)

To determine the unsteady lift for the system presented in Fig. 3.1, the velocity distribution induced by the chordwise distribution of vorticity must be converted into a pressure distribution via the unsteady Bernoulli equation (Eq. 2.12) as derived in Chapter 2. This equation, neglecting body forces, is

$$\frac{\partial \phi}{\partial t} + \frac{p}{\rho} + \frac{1}{2} |\nabla \phi|^2 = C(t)$$

(3.1)

where \(\phi\) is the potential, \(p\) is the pressure, \(\rho\) is the density, \(\nabla \phi\) is the velocity, and \(C(t)\) is a function of time only. The first term in this equation \(\left(\frac{\partial \phi}{\partial t}\right)\) accounts for apparent mass (non-circulatory) effects in the calculation of local pressures [38].
The velocity term accounts for circulatory effects through enforcement of flow tangency on the lifting surface. This includes the influence of circulation shed from the lifting surface in unsteady flow as discussed in Section 2.4.4.

The conversion of the circulation distribution to a force can be accomplished in a straightforward manner as described in Appendix B. The resulting force is oriented in the $\zeta$ direction as the pressure on a flat plate results in a force perpendicular to the plate. This force is given by

$$F'_\zeta = \rho \int_0^C \left[ \left( u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*) \right) \gamma_\xi + \frac{\partial \Gamma(\xi,t)}{\partial t} \right] d\xi$$  \hspace{1cm} (3.2)

The first term on the right hand side of Eq. 3.2 is derived from the circulatory terms in the unsteady Bernoulli equation and thus accounts for quasi-steady lift and the influence of circulation shed into the wake via the implementation of time dependent boundary conditions. The second term on the right hand side of Eq. 3.2 comes from the $\frac{\partial \phi}{\partial t}$ terms and thus accounts for non-circulatory effects. If the chordwise vorticity is collapsed into a single lifting line, the result is

$$F'_\zeta = \rho \left( u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*) \right) \Gamma + \rho \frac{\partial \Gamma}{\partial t} c$$  \hspace{1cm} (3.3)

where $c$ is the chord length.

At this point, it is necessary to introduce the unsteady Kutta-Joukowski approach. The vector form of the unsteady Kutta-Joukowski according to Drela [68] is

$$\overrightarrow{F_L'} = \rho \Gamma V_\infty \times \hat{s} + \rho \frac{\partial \Gamma}{\partial t} \frac{c}{|V_\perp|} V_\infty \times \hat{s}$$  \hspace{1cm} (3.4)

where
\[ V_{\perp} = V_\infty - (V_\infty \cdot \hat{s})\hat{s} \]  

(3.5)

and \( \hat{s} \) is the unit vector aligned with \( \Gamma \). If the velocity is assumed to be perpendicular to the lifting line (as in Fig. 3.1), Eq. 3.4 simplifies to

\[ F'_L = \rho \Gamma V_\infty + \rho \frac{\partial \Gamma}{\partial t} c \]  

(3.6)

A comparison between Eq. 3.3 and Eq. 3.6 reveals three important points. First, the apparent mass term is exactly the same in the two formulations indicating either that the two methods agree in their respective assessment of the apparent mass or that the unsteady Kutta-Joukowski approach uses the flat plate pressure integration result to account for apparent mass. Second, the circulatory lift terms are different in their assessment of the velocity both in magnitude and direction. This difference in magnitude is worth noting, as the magnitude of the integrated pressure force is necessarily smaller (albeit slightly) than the magnitude of the Kutta-Joukowski approach at any non-zero angle of attack. Finally, the resultant forces calculated here are in different directions. The lift force calculated with the unsteady Kutta-Joukowski is in the free-stream lift direction. The integrated pressure force is in the \( \zeta \) direction. The pressure force must be resolved into the free-stream lift direction via coordinate transformations (\( \alpha^* \) and \( \alpha^t \) in the Fig. 3.1), which would further reduce the magnitude of the force compared to that calculated with the Kutta-Joukowski.

The reduction in magnitude of the pressure integration force as compared to the Kutta-Joukowski force is attributable to the previously mentioned leading-edge suction force, which is accounted for implicitly within the Kutta-Joukowski approach [69] but is notably missing in the pressure integration approach for this
model. Its contribution is expected to be small in magnitude for lift [44] but non-negligible in induced drag, as discussed in Section 3.3. Further discussion of the leading-edge suction is provided in Section 3.4.

In summary, confidence in the unsteady Kutta-Joukowski approach to the calculation of lift has been provided through a term by term comparison with an unsteady pressure integration on a two-dimensional flat plate. This discussion also illuminated the source of the terms in the unsteady Kutta-Joukowski approach and the differences between the methods. Further discussion concerning the numerical properties of this approach (particularly with regard to the influence of the apparent mass term) is provided in Chapter 5.

### 3.2.2 Implementation

To clarify how the previous material is applied within the HOFW method to calculate lift, it is first necessary to identify the differences between a lifting surface modeled with DVEs and that described in Fig. 3.1. The section described in Fig. 3.1 features a continuous but otherwise unspecified distribution of vorticity in the chordwise direction. In contrast, the chordwise distribution of vorticity in the HOFW method consists of Dirac delta functions for each lifting line at a given spanwise location. These functions then integrate to step functions in the chordwise circulation distribution. A visualization of this concept is shown in Fig. 3.2.
Figure 3.2: Chordwise distribution of circulation at a given spanwise location as it relates to vorticity on the lifting surface represented with five rows of DVEs.

Application of Eq. 3.2 to a surface modeled with $m$ rows and $n$ columns of DVEs can be completed as follows, beginning with Eq. 3.2 integrated over the span of the wing:

$$
\bar{F}_c = -\rho \int_{-b/2}^{b/2} \int_0^c \left[ (u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*)) \cdot \gamma_c + \frac{\partial \Gamma(\xi, \eta, t)}{\partial t} \right] d\xi d\eta \tag{3.7}
$$

The chordwise and spanwise integrals can be converted into summations of the same integrals over each individual DVE, where the vorticity is estimated via a the change in integrated circulation (as visible in Fig. 3.2), viz.
\[
F_\zeta = -\rho \sum_{i=1}^{m} \sum_{j=1}^{n} \left[ \int_{n_{i,j}}^{n_{i,j}} \left( u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*) \right) \cdot (\Gamma_{i,j} - \Gamma_{i-1,j}) \, d\eta \\
+ \int_{n_{i,j}}^{n_{i,j}} \int_{-\xi_{i,j}}^{\xi_{i,j}} \frac{\partial \Gamma(\xi, \eta, t)}{\partial t} \, d\xi d\eta \right]
\]

(3.8)

The non-circulatory term (last term on the right hand side) requires special attention due to the nature of the DVE. As is evident from Fig. 3.2, the chordwise integrated circulation is constant for each DVE. Thus, the integration over each DVE is the DVE length multiplied by the circulation on that DVE.

\[
\int_{\eta_{i,j}}^{\eta_{i,j}} \int_{-\xi_{i,j}}^{\xi_{i,j}} \frac{\partial \Gamma(\xi, \eta, t)}{\partial t} \, d\xi d\eta = \int_{\eta_{i,j}}^{\eta_{i,j}} \frac{\partial \Gamma(\xi, \eta, t)}{\partial t} \left(2\xi_{i,j}\right) \, d\eta
\]

(3.9)

Substitution of the results from Eq. 3.9 into Eq. 3.8 results in the final equation for force in the \( \zeta \) direction,

\[
F_\zeta = -\rho \sum_{i=1}^{m} \sum_{j=1}^{n} \left[ \int_{n_{i,j}}^{n_{i,j}} \left( u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*) \right) \cdot (\Gamma_{i,j} - \Gamma_{i-1,j}) \, d\eta \\
+ \int_{n_{i,j}}^{n_{i,j}} \int_{-\xi_{i,j}}^{\xi_{i,j}} \frac{\partial \Gamma(\xi, \eta, t)}{\partial t} \left(2\xi_{i,j}\right) \, d\eta \right]
\]

(3.10)

The implementation approach has been described using a pressure integration approach to this point. Accordingly, the velocity terms in Eq. 3.10 include the freestream (kinematic) velocity and the velocity induced by the wake and other surfaces. The actual implementation uses a form of Eq. 3.4 for each DVE and separates the freestream (kinematic) force from the induced force, as the former can be calculated analytically and the latter must be integrated numerically.

Treating each DVE as an individual lifting line with Eq. 3.4, the freestream
(kinematic) normal force may be calculated as

\[
F_k = \rho \sum_{i=1}^{m} \sum_{j=1}^{n} \left[ \int_{-\eta_{i,j}}^{\eta_{i,j}} |V_\infty \times \hat{s}| \left( \Gamma_{i,j} - \Gamma_{i-1,j} \right) d\eta 
+ \int_{-\eta_{i,j}}^{\eta_{i,j}} \frac{\partial \Gamma(\xi, \eta, t)}{\partial t} (2\xi_{i,j}) d\eta \right] \left( \frac{V_\infty \times \hat{s}}{|V_\infty \times \hat{s}|} \right)
\]

(3.11)

where \(\hat{s}\) is the unit vector in the global reference frame along the leading edge bound vortex of the DVE. In this way, the local lift of each DVE is resolved in the global reference frame such that the contributions can be summed.

The forces due to induced velocities are calculated similarly using the local induced velocity instead of the freestream (kinematic) velocity, viz.

\[
F_{ind} = \rho \sum_{i=1}^{m} \sum_{j=1}^{n} \int_{-\eta_{i,j}}^{\eta_{i,j}} |V_{ind} \times \hat{s}| \left( \Gamma_{i,j} - \Gamma_{i-1,j} \right) d\eta \times \left( \frac{V_\infty \times \hat{s}}{|V_\infty \times \hat{s}|} \right) \cdot \left( \frac{V_{ind} \times \hat{s}}{|V_{ind} \times \hat{s}|} \right)
\]

(3.12)

The induced velocities can be calculated at any location within the flow field based on the strengths of all singularities within the solution and their proximity and orientation relative to the location of interest. As can be surmised from the calculation of velocities induced by a single DVE described in Appendix A, this process requires several calculations per DVE. As the number of DVEs increases, the number of calculations increases as well such that it can become unwieldy to calculate the velocity at many locations on each DVE. As a compromise between computational efficiency and solution accuracy, the induced velocities are calculated at three spanwise locations along the leading edge of each DVE and the resulting in-
duced lift is integrated using Simpson’s rule as recommended by Horstmann [43]. Because only the forces in the local lift direction are desired from this calculation, the resulting forces are then projected onto the local lift direction.

The total force due to the lift of each local section is the combination of Eqs. 3.11 and 3.12, viz.

\[ F_{\text{total}} = F_k + F_{\text{ind}} \]  

(3.13)

In non-planar wing cases, it is noteworthy that these total forces calculated as lift may have components in the body-fixed coordinate frame side-force directions. In addition, on a rotating surface such as a propeller, the local lift and drag directions will vary considerably from the body-fixed coordinate system, and from time-step to time-step.

### 3.3 Vortex-Induced Drag

As with the lift force, the vortex-induced drag is physically a result of pressures on the surface. Unfortunately, the use of surface pressure integration to calculate this force in potential flow methods has been shown to be ineffective, mostly due to the inability of the methods to accurately resolve the pressure distribution about the leading edge of the airfoil [70]. This is particularly true of the HOFW model of a lifting surface, as it does not account for the thickness of the lifting surface. To account for the pressure distribution about the leading edge in this model, an ad-hoc estimation of the leading-edge suction force is necessary (as discussed further in Section 3.4) and carries with it the potential for additional error.

An alternative to surface pressure integration is an application of the Kutta-
Joukowski theorem. In this approach, the induced drag is calculated as the cross product of the downwash induced by the trailing vortices on the lifting line with the circulation at that spanwise location. This approach can be proven valid for a lifting line with a fixed, drag-free wake through Trefftz plane analysis. An example of such a proof is provided in Appendix C.1 along with the extension of the concept for multiple wing-wake systems in Appendix C.2.

In the classical approach, the lifting line is located at the quarter-chord location of the lifting surface for this calculation. The “downwash” calculated at this location is non-physical in that it would indicate a flow through the wing surface. This procedure additionally neglects the influence of the trailing-edge shape. The trailing edge shape is important because it is the location at which the vorticity is physically shed into the flow field and thus has an impact on the wake shape, and consequently, the wake-induced velocities \cite{71,72}. An alternative is to place the lifting line at the trailing edge of the wing for the calculation of drag, such that the wake shape reflects the trailing edge location \cite{72,73}.

Wake relaxation, as is present in the HOFW method, introduces additional concerns. If the wake is relaxed rather than fixed and drag-free, the classical Trefftz plane technique for verifying the application of the Kutta-Joukowski law at the trailing edge becomes impractical for two main reasons. First, it is difficult to identify the Trefftz plane itself, as the trailing vortices rarely (if ever) reach an equilibrium condition. Second, one cannot conduct a path integral into the cores of the rolled-up tip vortices. Eppler and Schmid-Göller \cite{72} provide qualitative support of this application with the assertion that because all of the wake energy induced by a lifting surface is present at its trailing edge, it can be assumed that the induced drag can be calculated at this location (rather than in the Trefftz plane)
without causing significant errors. This approach is also supported by successful applications for both fixed [8, 45, 60, 74] and rotating lifting surfaces [46, 56, 61, 62] and through direct comparisons with other methods [71, 75]. As a result and for lack of a better alternative, the HOFW method employs the Kutta-Joukowski approach at the trailing edge for calculation of vortex-induced drag.

Proving the applicability of this approach for propeller-wing systems elicits a new set of challenges. The propeller adds energy to the flow that must be considered. The periodic, unsteady nature of the flow-field about this wing must also be addressed. In light of these challenges, a re-examination of the influence of wake relaxation is also warranted.

To address these concerns, the approach outlined in Fig. 3.3 is taken. The first step (outlined in green) was addressed earlier within this section. The next step (outlined in blue) is addressed in Section 3.3.1 and accounts for energy added to the system by the propeller through a Trefftz plane analysis with a fixed wake. This analysis also addresses the unsteady nature of the flow-field. Next, because the Trefftz plane analysis assumes a fixed wake, a connection must be made between calculation of the induced drag with a fixed, drag-free wake and that with a relaxed wake. Although a conclusive proof is not provided, a qualitative argument for agreement between these two models with respect to induced drag is discussed in more detail in Section 3.3.2.

### 3.3.1 Trefftz Plane Analysis

In this section, a full Trefftz plane analysis of a propeller-wing system is discussed. The goal of this section is to directly address the energy added to the system by the propeller and to discuss the influence of unsteadiness. Although the analysis herein
Figure 3.3: Flow chart outlining justification for use of the Kutta-Joukowski law in calculation of induced drag for propeller-wing systems with relaxed wakes.
utilizes fixed-wake assumptions, the influence of wake relaxation is addressed in Section 3.3.2.

### 3.3.1.1 Flowfield Unsteadiness

The interaction between the propeller and the wing can be assumed to be periodic-unsteady for the purposes of control volume analysis according to Drela [69]. The periodic-unsteady velocity components \(( \tilde{u}, \tilde{v}, \tilde{w} )\) can be expanded about their mean values \(( \bar{u}, \bar{v}, \bar{w} )\) as

\[
\tilde{u}(x, y, z, t) = \bar{u}(x, y, z) + \sum_{k=1}^{\infty} u_k(x, y, z) \sin \frac{2\pi k t}{t_p} (3.14)
\]

for example, where \(t_p\) is the period. Thus, time averaging the velocities results in

\[
u(x, y, z) = \frac{1}{t_p} \int_0^{t_p} \tilde{u} dt = \bar{u} \quad (3.15)
\]

In the case of the product of velocities, such as \(uv\), additional terms exist. These terms are expected to be negligible for propeller-wing systems according to Drela [69].

Accordingly, the Trefftz plane analysis that follows addresses the time-averaged flow field. The propeller is treated as an actuator disk that induces velocities in both the axial and tangential directions. These velocities are assumed to be time-averaged velocities of a fixed-wake HOFW modeled propeller.

### 3.3.1.2 Discussion of Analysis

A conceptual discussion of the assumptions and results of the Trefftz plane analysis is provided in this section. The full details are provided in Appendix D for reference.
The propeller-wing system for this analysis is reduced to a time-averaged propeller wake (including induced velocities in the axial and tangential directions) and a downstream wing with a fixed, drag-free wake. The energy added to the flow by the propeller is accounted for through balancing of the thrust and torque in the Trefftz plane, revealing the losses due to the interaction between the two systems. These losses, in agreement with Veldhuis [3], are given in the Trefftz plane as

\[ D_{\text{int}} = \rho \int_{-b/2}^{b/2} \Gamma_w(y) V_{p,n} dy \]  \hspace{1cm} (3.16)

where \( b \) is the span of the wing, \( \Gamma_w(y) \) is the circulation distribution of the wing, and \( V_{p,n} \) is the velocity normal to the wing wake in the Trefftz plane due to the propeller. It is important to note that these losses are in addition to the swirl losses due to the propeller wake rotation, as swirl losses were taken into account through balancing of the propeller torque.

If this result is combined with the induced drag due to the wing in the Trefftz plane, the result is

\[ D_t = \rho \int_{-b/2}^{b/2} \Gamma_w(y) \left( V_{p,n} + \frac{V_{w,n}}{2} \right) dy \]  \hspace{1cm} (3.17)

where \( V_{w,n} \) is the downwash induced in the Trefftz plane by the doubly infinite planar vortex sheet. At the trailing edge, the downwash induced by the wake is half due to the fact that the infinite vortex sheet in the Trefftz plane is semi-infinite at the trailing edge of the wing (as further discussed in Appendix B). The normal velocities due to the propeller wake are constant in this model (and in agreement with the physical system) as the tangential induced velocities are constant with downstream distance. Thus, the total induced drag can be calculated at the trailing edge as
\[ D_i = \rho \int_{-b/2}^{b/2} \Gamma_w(y) (\nabla_{p,n,te} + \nabla_{w,n,te}) dy \] 

(3.18)

This result is agrees with an application of the Kutta-Joukowski law at the trailing edge of the wing that includes the velocities induced by the propeller.

The model used in this analysis differs from the HOFW model in that the the interference drag losses in the HOFW model are actually split between the propeller and wing. This manifests in the fact that the local induced drag calculation in the HOFW method only accounts for the influence of the wakes (and not lifting surfaces). The tangential induced velocities by the propeller are constant with downstream location due to a combination of the bound vorticity and trailing vorticity (as discussed in Section 1.3.1.1). The velocity induced by the propeller wake alone on a downstream wing may be slightly less than that in the Trefftz plane depending on the distance between the two systems. This deficit is made up by a slight loss in thrust due to the velocities induced by the downstream propeller wake. The limits of this statement are addressed by Kroo [34], who showed that the interference drag of an upstream propeller with a downstream wing is equivalent to the interference thrust of an upstream wing and downstream propeller.

3.3.2 Relaxed Wakes vs. Fixed, Drag-Free Wakes

The discussion in Section 3.3.1 assumes fixed, drag-free wakes for the wing and propeller. To apply these results to a model with relaxed wakes it is necessary to discuss the similarity between the calculation of induced drag with fixed, drag-free wakes and that with relaxed wakes.

A rigorous proof showing that the calculation of induced drag on a lifting surface with a relaxed wake is equivalent to that on a lifting surface with a fixed, drag-
free wake with the same circulation distribution would be ideal, as it would show that the results of Section 3.3.1 apply directly to a relaxed-wake system. Schmid-Göller [70] approached this result by proving that the induced drag of a wing with a relaxed wake is equal to that calculated with the trace of the rolled-up wake in the Trefftz plane as the lifting line appended with a fixed, drag-free wake. Smith [76] showed that as long as deflection angles and curvature in the wake are small, a drag-free wake may be substituted for a relaxed wake at the trailing edge of the lifting surface without inducing significant error in induced drag. Unfortunately, a full proof of equivalency between these systems has remained elusive.

In lieu of such a proof, evidence and qualitative support for the use of the Kutta-Joukowski applied at the trailing edge for induced drag calculation (as discussed in the beginning of Section 3.3) can be used to support a similarity between these two systems. To make this argument, the following four statements are of use:

(a) The wake(s) are modeled as fixed and drag-free.

(b) The wake(s) are modeled as relaxed (force-free).

(c) The Kutta-Joukowski law applied at the trailing edge is defensible for calculation of induced drag.

(d) The Kutta-Joukowski law applied at the trailing edge is defensible for propeller-wing systems.

If it is assumed that the Kutta-Joukowski as applied at the trailing edge is only rigorously defensible for the case of a fixed, drag-free wake (as seems to be the case) then the following argument can be made:
In words, this argument can be summarized as follows. The Kutta-Joukowski law applied at the trailing edge is rigorously defensible for calculation of induced drag if and only if the lifting surface is modeled with a fixed, drag-free wake \((c \Leftrightarrow a)\). Despite lacking a rigorous proof, evidence suggests that this approach also works for a relaxed wake \((b \rightarrow c)\). Thus, the properties relating to the induced drag of a relaxed wake are implied to be consistent with those of a fixed wake \((b \rightarrow a)\). If this is the case and the Kutta-Joukowski approach is valid for a fixed, drag-free wake model of a propeller-wing system \((a \rightarrow d)\), then it is also valid for a relaxed wake model \((b \rightarrow d)\).

A few caveats to this result must be mentioned. First, in order for this argument to be true, the two lifting surfaces must have the same circulation distribution. It is demonstrably false that the induced drag of a system in which the flow tangency has been enforced while incorporating wake relaxation effects is necessarily the same as one in which flow tangency has been enforced using a fixed, drag-free wake. Second, this argument does not prove that a fixed, drag-free wake is equivalent to a relaxed wake. It implies only that evidence suggests that they are reasonably consistent for the purposes of calculation of induced drag. This conclusion is supported by the reasoning of Eppler and Schmid-Göller [72] in that in both models, all of the energy of the wake due to the lifting surface is present at the trailing edge location.
Although it was addressed in the Trefftz plane analysis, a revisiting of energy addition due to the propeller may be warranted in the context of relaxed wakes. This energy is accounted for in the model through velocities induced by the rotating lifting surfaces and their wakes. In this respect, the propeller-wing system is equivalent to an interacting multiple wing-wake system as has been specifically addressed both in Appendix B.2 for fixed, drag-free wakes and in the work of Bramesfeld [60] for relaxed wakes.

3.3.3 Summary of Support of Approach to Induced-Drag Calculation

A summary of the information contained in this section can be accomplished in three steps:

1. The application of the Kutta-Joukowski law for induced-drag calculation was proven through a Trefftz-plane analysis that used fixed, drag-free wakes for both single and multiple wing-wake systems.

2. The application of the Kutta-Joukowski law for induced-drag calculation for propeller-wing systems was proven through a time-averaged Trefftz plane analysis that used fixed, drag-free wakes. Time averaging accounted for the periodic-unsteady flowfield introduced by the interaction between the propeller and wing.

3. Given qualitative and experiential support of the use of the Kutta-Joukowski law at the trailing edge for calculation of induced drag with relaxed wakes along with the previous two statements, a logical argument was made for
the validity of use of the Kutta-Joukowski law at the trailing edge for the
calculation of induced drag in propeller-wing systems with relaxed wakes.

3.3.4 Implementation

This approach is implemented within the HOFW method in three steps. First, if
the trailing edge is swept it must be “deswept”. This is necessary because a swept-
wake vortex sheet is singular along its leading edge [43], and thus an application
of the Kutta-Joukowski theorem results in infinite vortex-induced drag. It is valid
to locally straighten the trailing edge because it has been analytically proven that
the vortex-induced drag of an unswept system has equivalent induced drag to that
of a swept system with the same spanwise circulation distribution [37]. The second
step is to calculate the drag force is via the Kutta-Joukowski theorem at the mid-
span and \( \pm 0.8\eta \) locations as recommended by Bramesfeld [8] and later Basom [46].
The bound vorticity used for this calculation is the sum of the bound vorticity
on the lifting lines at that location. The normal velocity is that induced by the
wakes of all lifting surfaces. Finally, the drag at these three locations is numerically
integrated via Simpson’s rule to determine the total drag on the surface.

3.4 Consideration of the Leading-Edge Suction

Force

The leading-edge suction force is the force that results from the pressure distribu-
tion about the leading-edge radius of an airfoil of finite thickness. In potential flow,
the two-dimensional pressure drag on an airfoil must be zero. The leading-edge
suction force in a potential flow is thus equal and opposite of the drag force due
to the integrated pressure aft of the leading-edge region.

If pressure integration is used with a flat plate representation of a lifting surface for the calculation of lift and drag, the leading-edge suction force is not physically represented and must be added. In this case, the force itself must be tangent to the flat plate (as normal forces are accounted for through pressure on the plate) and, as a result, has a small component in the lift direction as well.

In the HOFW method, the forces are not calculated through pressure integration but with an application of the Kutta-Joukowski law, as described in Section 3.2.2 for lift and Section 3.3.4 for drag. In this approach, there is no two-dimensional pressure drag and thus any accounting for the leading-edge suction force in the drag calculation is unnecessary. This conclusion is supported by Drela [77], who states in reference to a vortex-lattice method:

“Interestingly enough, the Kutta-Joukowsky force calculation form gives exactly zero drag in the 2D case where there are no trailing [horse-shoe vortex] legs. Therefore it implicitly accounts for the leading edge suction force, which then does not need to be added explicitly.”

It is worth mentioning that the addition of profile drag through look-up tables accounts for any reduction in leading-edge suction due to viscosity as is accounted for explicitly by Leishman [78] when discussing two-dimensional unsteady airfoil drag. Further comparison of the HOFW method to the approach recommended by Leishman [78] is provided in Section 3.5.
3.5 Profile Drag Estimation

In a manner similar to that of Maniaci [56], profile drag is taken into account by splitting each lifting surface into chordwise strips which are presumed to act as two-dimensional airfoils. Experimental or externally calculated airfoil data, including the profile drag coefficients at a set of Reynolds numbers and lift coefficients, is then used to estimate the profile drag under local conditions. This process is summarized in Fig. 3.4. For each spanwise location, the local lift coefficient is calculated as the ratio of the sum of the forces on the SDVEs along the chord resolved in the local lift direction and the local dynamic pressure. The local Reynolds number is calculated with the local velocity and chord of the section. The local profile drag coefficient is calculated with a two-dimensional interpolation of the airfoil data based on the Reynolds number and lift coefficient, and dimensionalized to profile drag using the local dynamic pressure. The profile drag is presumed to act in the direction of the local fluid velocity, and resolved in the global coordinate frame appropriately.

![Figure 3.4: Definition of velocities and forces for the calculation of profile drag.](image)

This process relies on an accurate evaluation of the local velocity. The local
velocity, $V_{lavg}$ as defined in Fig. 3.4, is the vector sum of the kinematic velocity (including the rotational velocity for rotating surfaces) and the induced velocities. The induced velocities are particularly important in estimation of the profile drag in propeller-wing systems because the lift coefficient of any section of a lifting surface operating within the propeller slipstream will be smaller due to the inclusion of the axial induced velocities of the propeller. This reduced lift coefficient will then influence the predicted drag coefficient, which is again dimensionalized with the local velocity. Within this method, the induced velocities are calculated at the center of the leading edge of each SDVE along the lifting surface’s leading edge.

In the case of multiple lifting lines in the chordwise direction, the induced velocity at the center of the leading edge is averaged with the mid-chord induced velocity of the SDVE and the upstream SDVE to dampen any errors due to discontinuities in the bound vortices of a surface with twist. The induced velocities for each SDVE calculated in this manner are then averaged over the chord.

Because the local lift and drag direction are different on each strip of the lifting surface due to variations in induced velocities, the calculated profile drag is transformed into components in the global coordinate frame. In this way, it can be combined with the lift and vortex-induced drag forces, and the net forces can be resolved into the appropriate reference frame. For example, the forces on the wing are traditionally resolved into lift and drag, which are defined as perpendicular and parallel to the true free-stream velocity. The forces on a rotating surface are typically resolved into thrust and torque, as is further explained in Section 3.7.

To discuss the influence of two-dimensional unsteadiness on profile drag, it is helpful to compare the current method with the approach recommended by Leishman [78]. In this model, the steady airfoil profile drag is given by
\[ c_d = c_{d,0} + c_{d,P} \]  
(3.20)

where \( c_{d,0} \) is the viscous drag and \( c_{d,P} \) is the pressure drag. The pressure drag is further broken down as

\[ c_{d,P} = c_n \sin \alpha - \eta c_c \cos \alpha \]  
(3.21)

where \( c_n \) is the normal force, \( c_c \) is the leading-edge suction coefficient, and \( \eta \) is a correction for the lack of 100% leading-edge suction attained in real flows (as compared with potential flows). Although not explicitly stated in the reference, the description agrees with an assessment of the forces on a flat plate potential flow model in which forces are calculated through pressure integration and then corrected for viscous effects. All of the terms in Eqs. 3.20 and 3.21 are thus accounted for in the HOFW model through the profile drag look-up tables.

The influence of unsteadiness is taken into account by Leishman [78] using an indicial method. Omitting the details for brevity, the final form of the unsteady profile drag is given as

\[ c_d(t) = c_{d,0} + c_n(t) \sin \alpha(t) - \eta c_c(t) \cos \alpha(t) \]  
(3.22)

where \( \alpha(t) \) is the airfoil (geometric) pitch angle, the unsteady leading-edge suction term \( (C_c) \) depends only on the circulatory lift, and the unsteady normal loading term \( (C_n) \) includes non-circulatory effects. Alternatively, the HOFW method calculates profile drag in unsteady analysis at each time-step as

\[ c_d(t) = c_d(c_l(t), Re(t)) \]  
(3.23)
where $c_l(t)$ includes circulatory and non-circulatory effects. This drag is applied along the local aerodynamic velocity vector.

Because of the inherent differences in these two approaches, it is difficult to compare terms on a one to one basis. Still, one conclusion can be drawn with regard to the unsteady leading-edge suction term. The approach recommended by Leishman decouples the leading-edge suction force and the remaining pressure drag whereas the table look-up approach combines these terms. In other words, in the HOFW method, the leading-edge suction force is included in the profile drag, which varies with the total lift coefficient (including circulatory and non-circulatory effects). The leading-edge suction term in Leishman’s approach varies only with circulatory effects.

Because the approach to calculation of induced-drag within the HOFW method is defensible only in the time-averaged sense, a comparison of the total time-accurate drag results with other methods is not meaningful. As a result, further consideration of this difference and the influence of leading-edge suction on the time-accurate profile drag has been delegated to future work.

### 3.6 Stall Model

A simple stall model was introduced to identify sectional stall conditions and predict the lift of the wing as it approaches the stall condition. This model is based on the approach of McCormick [25]. The local lift coefficient (calculated in the manner described in the previous section) may not exceed the maximum two-dimensional lift coefficient ($c_{l,max}$). Thus, if a section’s lift coefficient exceeds the maximum lift coefficient from the look-up table, the section is considered “stalled” and held at that maximum lift coefficient and profile-drag coefficient. The resulting forces are
then decomposed into components in the body-fixed reference frame.

There are several reasons why this approach is not adequate for performance prediction in stalled conditions and is only recommended for identification of local stall. First, this is a post-processing step and as such, does not influence the actual flow field solution. The flow field continues to develop as if the flow remained attached. Thus, the induced drag continuous to increase as if the wing has not stalled and the local velocities become less and less realistic. Second, the model does not account for any non-linearity in the two-dimensional lift curve. As such, the lift curve stays constant at $2\pi$ per radian until the stall condition is reached, at which point it remains zero rather than decreasing as would be seen in a true two-dimensional stall. That said, because different sections of the wing stall at different times, the non-linearity of the three-dimensional lift curve as it approaches the stall condition may be approximated (albeit crudely). Finally, the actual two-dimensional profile drag in stall conditions increase significantly rather than stay constant as in the model.

### 3.7 Propeller Performance

Once the resultant force (including both lift and drag) is determined for each panel in a specified direction in the global reference frame, it can be decomposed into a thrust component and a torque component for the rotating lifting surfaces. The relevant velocities and forces as they are defined relative to the lifting surface are shown in Fig. 3.5. The thrust direction is defined in the direction of the axis of rotation. The torque direction for a particular SDVE at an instant in time is defined as the vector in the plane of rotation perpendicular to the vector from the axis of rotation to the SDVE. These definitions are standard for thrust and
torque [25].

Figure 3.5: Definition of velocities and forces on a section of a rotating lifting surface.

In addition to thrust and torque, the propeller also produces a normal force when inclined relative to the free-stream velocity [3, 79, 80], as depicted in Fig. 3.6. This force can be extracted from the local lift and drag forces on each section simply by resolving the components of the local forces in a defined propeller normal force direction.
The relative contribution of the propeller normal force to the integrated forces depends on system, but is typically small. As an example, a simple propeller-wing system was analyzed using the semi-empirical method recommended by Veldhuis [3] at three propeller pitch settings. The zero lift angle of attack was assumed to be inclined $5^\circ$ relative to the thrust line, and the wing was assumed to have a lift curve slope estimated based on its aspect ratio. No mutual interaction was considered for a conservative estimate, as the induced velocities from the propeller would increase the lift on the wing and reduce the relative contribution. Even with this assumption, the results show that the normal force contributes less than 1.2% of the total lift force as shown in Fig. 3.7. Because the all cases considered within this thesis require propeller inclination angles relative to the free-stream of less than $10^\circ$, this force was neglected and a thorough implementation and investigation of the force was left as future work.
Figure 3.7: The propeller normal force as a percent of the total integrated force in the lift direction for a simple propeller-wing system.

3.8 Thrust and Drag Bookkeeping

In the design of aircraft systems, it has historically been necessary to establish a thrust-drag bookkeeping system to parse out the influence of interactions between components at various operating points in the performance of the vehicle. According to Covert et al. [81],

"The need for a bookkeeping system in the performance prediction phase on an aircraft development program arises largely from the inability to determine the performance of the complete airplane system, with simultaneous real inlet and an exhaust system operation, in a
single test or calculation.”

For example, Covert et al. defines a force balance as follows:

\[ F_{EX} = F_N + \Delta F_{INL} + \Delta F_{EXH} + \Delta F_{TRIM} \]
\[ - D_{REF} - \Delta D_{INL} - \Delta D_{EXH} - \Delta D_{TRIM} - \Delta D_{RN} \]  

(3.24)

where \( F_{EX} \) is the total force imbalance between thrust and drag in the flight direction, \( F_N \) is the installed engine thrust at the reference condition, and \( D_{REF} \) is the external force predicted by the aerodynamic model at the reference condition. The remaining thrust terms (\( \Delta F \) terms) are throttle-dependent adjustments in the thrust due to the difference between the operating condition and the reference condition (\( INL \) referring to changes in inlet conditions, \( EXH \) referring to changes in exhaust conditions, and \( TRIM \) referring to changes in control surface positions). Similarly, the remaining drag terms (\( \Delta D \) terms) are adjustments in the drag due to the difference between the reference condition and actual operating conditions.

The HOFW method creates a flow solution that includes the influence of all lifting surfaces at all user-specified operating conditions. The thrust and drag (as well as lift and torque) are then calculated as described in previous sections based on that flow solution. As a result, it is unnecessary to correct the thrust and drag from a reference condition to the operational condition, as it is fast and straightforward to calculate the installed thrust and drag at each condition.

Despite this capability, it may be necessary to breakdown the thrust and drag calculated using the HOFW method into components for comparison with flight test data or for application of design methods. This can be accomplished through careful manipulation of operational conditions to isolate the desired terms. For
example, to investigate the change in propeller efficiency due to installation effects, the propeller performance as calculated within the propeller-wing system can be compared with that calculated for the same propeller modeled in isolation.

For assessment of overall vehicle performance and identification of trim conditions, the aggregate lift, drag, thrust, and torque are needed. Although the calculation of the forces on each SDVE has been explained previously, their contributions to the aggregate vehicle performance have not been addressed. The lift of the vehicle is the sum of the forces calculated on each SDVE on fixed (non-rotating) surfaces resolved in the free-stream lift direction (perpendicular to the free-stream and body-fixed y-axis). Likewise, the drag is the sum of the forces (including profile drag) calculated on each SDVE on fixed (non-rotating) surfaces resolved in the free-stream drag direction (parallel to the free-stream). The thrust is the sum of the forces on rotating surfaces resolved in the thrust direction (along the axis of rotation) and the torque is the sum of the torques calculated on each individual rotating surface.

3.9 Limitations of the Model

There are limitations in the similitude of the DVE representation of propeller-wing systems and real world propeller-wing interaction. These limitations are a result of the fact that the HOFW method is a mathematical model of a physical system that relies on many simplifying assumptions to make the problem more tractable. While significant effort has gone into identifying and clarifying all assumptions applied throughout the derivation process, a top-level assessment of deviations from the physics of real world systems is required as well.

A major concern with the method stems from the intersection of the propeller
wake with the wing surface, an example of which is shown in Fig. 3.8. The propeller wake DVEs are coincident with the wing SDVEs in two places within the figure, with the downstream location highlighted. This is disconcerting for two reasons. First, if the SDVE is representing a thin impervious surface, the vorticity could not physically extend through it as there would be no velocity to advect the wake in the plane of the surface. It would either need to split, with half of the wake traversing the upper surface and half traversing the lower surface, or it would need to fold over the leading edge, with part of the wake pinned to the leading edge stagnation point, and the rest of it stretching in the downstream direction. In addition, the vortex sheet of the wake DVE induces a shear layer with tangential velocities in opposite directions above and below the sheet. Thus, the second cause for concern is that flow tangency is not being enforced everywhere on an SDVE that is impinged upon by the wake DVE.

To address these concerns, it is worthwhile to first consider qualitatively the physical interaction of the wake and the surface. Johnston and Sullivan [5] used smoke flow to visualize the location and deformation of a propeller tip vortex as it traversed a lifting surface under various conditions. A visualization of one of the findings of this study is reproduced in Fig. 3.9. The tip vortex splits over the lifting surface, and the upper and lower portions of the tip vortex follow slightly different paths. The authors attribute this behavior to either the spanwise wing pressure gradient or an image vortex effect. The two pieces then reconnect at the trailing edge with a slight misalignment. The size of this misalignment depends on the angle of attack of the wing, with increasing misalignment at increasing angles of attack.
Figure 3.8: Image of the intersection of the wake and the wing as exists within the HOFW method. The wing and propeller lifting SDVEs are shown in red and the wakes are shown in blue. One location of intersection is highlighted with a gold oval.
Referring to the first concern with the HOFW method listed above in light of the qualitative behavior of the tip vortices, if it is assumed that the inner wake behaves in accordance with the tip vortices, it follows that splitting the wake around the lifting surface would be most in line with the physical system. Although simple in concept, practical implementation of this wake-splitting approach within the HOFW method is less straightforward. According to the third theorem of Helmholtz [67], the vorticity cannot begin or end within the fluid. It must either be a closed loop or begin/terminate on a solid boundary. The latter option is consistent with the flow patterns shown by Johnston and Sullivan [5], where the vorticity seems to terminate on the solid surface until it reaches the trailing edge and reconnects. To model this in the potential flow domain, however, would violate the governing equations of potential flow, as it would require a vortex to begin/end in a fluid.

Two practical and potential-flow consistent options exist to solve this problem.
One option is to split the wake DVE around the SDVE such that the upper surface and lower surface wakes are essentially independent closed loops, as sketched in Fig. 3.10. This would result in wake DVE edges on the upper and lower surface with the singular tip-vortices that accompany them. The induced velocity field in this model is much different from that resulting from a vortex sheet that terminates on and is forced to remain attached to the lifting surface.

![Figure 3.10: Sketch of the model that would result from splitting the wake around the wing in the HOFW method.](image)

The simple alternative and second option is to allow the wake DVE to impinge upon the SDVE. Because the SDVE is infinitely thin, this option is a fairly good
approximation to terminated upper and lower surface vorticity, with the exception of the ability of that vorticity to move independently. The difference in velocities on the upper and lower surface can be accounted for in the wake shape with sufficient discretization. This issue is further explored in Chapter 4 through resolution studies. Despite the fact that the HOFW method does not model the physical splitting of the wake, it is worth noting that the method predicts wakes that are qualitatively similar to those described by Johnston and Sullivan [5].

The concern about flow tangency on the lifting surface at locations of intersection between the propeller wake and wing is another consequence of the simplified mathematical model used to represent a more complex physical system. Before embarking on a more detailed explanation, it is worthwhile to dispense a common misconception concerning the wake as it relates to streamlines. One reason that it appears that flow tangency is not enforced in the method is that the wake vorticity impinges upon the surface of the wing. In steady flow, the wake vorticity by definition is aligned with the flowfield (see Fig. 2.7), such that the vorticity and streamlines are coincident. As described in Section 2.4, this is not necessarily the case in unsteady flow, as the vorticity can also convect with the local velocity (see Fig. 2.8). Thus, the fact that the wake impinges upon the wing does not independently indicate a lack of flow tangency.

To address this concern further, it is first necessary to consider flow tangency on a fixed wing surface within the HOFW method. The flow tangency boundary condition is enforced at the control points. At locations on the lifting surface away from the control points, there is no explicit mechanism in the HOFW method by which flow tangency is enforced. Thus, for example, if the lifting surface is represented by one SDVE, flow tangency will occur at the one control point, but
will likely not occur elsewhere on the surface. The more control points that exist on
the surface, the more locations where flow tangency is enforced and thus, the more
the mathematical representation of the flow field resembles the expected physical
flowfield.

As evidence, streamlines for a single planar wing \((AR = 2.67)\) at a central
spanwise control point station are provided in Fig. 3.11. The number of chordwise
SDVEs within the figure increase from one in (a) to five in (d). With one SDVE, a
streamline can be seen clearly passing through the wing near the leading edge and
following a curved path slightly below the surface. In (b), two control points seem
to straighten the more aft streamlines, but flowfield does not look as smooth as it
does with three SDVEs, as shown in (c). Finally, with five SDVEs, it appears flow
tangency is fairly well approximated over the entire surface. Despite these changes
in the streamlines (and the low aspect ratio wing), the integrated lift coefficient
increases by only 1.6% from a single chordwise SDVE to five chordwise SDVEs.
Likewise, the induced drag coefficient increases by 3.4%.

Figure 3.11: Stream-lines (shown in blue) over the wing (shown in red) and wake
(shown in yellow) calculated at a single time-step of a single wing case with a.)
one row, b.) two rows, c.) three rows, and d.) five rows of surface DVEs.
Similar changes in streamline behavior can be seen in propeller-wing interaction cases with the additional influence of the propeller wake. For example, streamlines at a single time-step are plotted for three different resolutions in Fig. 3.12. This is the same case as is shown in Fig. 3.8, and the spanwise slice is taken where the downstream intersection occurs. With a single control point in (a), the downstream intersection does result in some through flow around the 3/4 chord point, but the main flow tangency issue is still near the leading edge (in the same location of the single wing). With two control points in (b), the flowfield near the trailing edge varies significantly from that predicted with a single control point. This is because the aft control point is near the propeller wake intersection and, as a result, the circulation on the aft SDVE is adjusted to compensate. Without an additional control point closer to the trailing edge to re-adjust the flowfield, the flow around the trailing edge does deviate some from what might be expected. That said, the adjustment results in an increase or decrease in lift on the aft SDVE, as would be expected due to the presence of pressure variations within the propeller slipstream (see Figs. 1.17 and 1.19). So again, while the flow field may vary from what would be expected, the forces are not necessarily in error, as is further explored in the resolution study conducted in Chapter 4. With three SDVEs in (c), the streamlines conform nicely to the lifting surface again.
Figure 3.12: Stream-lines (shown in blue) over the wing (shown in red) calculated at a single time-step of propeller-wing interaction with a.) one row, b.) two rows, and c.) three rows of surface DVEs. The wake DVEs (for both the wing and propeller) are shown in yellow.

Two things can be concluded based on this discussion. First, in the HOFW method, flow tangency is not enforced at all locations on the wing, and in the case of a single fixed wing, this behavior does not have a significant influence on forces. Second, while the intersection of the propeller wake on the wing does change the flowfield and, as a result, the chordwise lift distribution, the resulting streamlines are not overly unrealistic, and as shown in Chapters 4-7, the resulting time-averaged forces do converge to reasonable values.

3.10 Summary

In this chapter, the method for calculating lift and drag on each lifting surface has been explained. The lift is calculated using an unsteady version of the Kutta-Joukowski law on each lifting line. The drag is calculated using the Kutta-Joukowski law as well at the trailing edge. These approaches have been shown to be applicable to propeller-wing systems. Profile drag has been accounted for through the use of strip-theory and look-up tables. All forces on the rotating lifting surface
(propellers/proprotors) are resolved into thrust and torque, whereas the forces on the fixed lifting surfaces (wings) are resolved into lift and drag. Although there are limitations with respect to the models representation of the physical system in terms of the intersection between the wing and propeller wake, these limitations have been shown to be acceptable within the context of low-order models for design applications.
4.1 Overview

Within this chapter, the influence of the spacial and temporal resolution used to model a representative propeller-wing system on time-averaged predictions of integrated lift, drag, thrust, and torque are investigated. The purpose of this investigation is two-fold: first, to establish guidelines for the minimum resolution needed to capture the physics of the system for a reliable estimate of the integrated forces, and second, to provide bounds on the error that can be expected due to changes in resolution. To meet these goals, a discussion of the convergence properties of vortex-lattice methods is provided. Within this context, a study of the influence of grid resolution is provided, beginning with a description of the system and baseline resolution parameters. Each parameter is varied individually with the remaining parameters at their baseline levels, and the influence of that parameter on the integrated forces is shown.

It is important to note that the results for the propeller-wing system examined in this section are strictly valid only for systems of similar loading and relative size.
of the components in quasi-steady analysis. The influence of grid resolution on
time-accurate (both quasi-steady and unsteady) analysis is addressed in Chapter
5. In light of these restrictions, the minimum resolution determined in this study is
recommended as a starting point for future studies. That said, similar convergence
characteristics were found for the application studies within this thesis.

4.2 Grid Resolution Properties of

Vortex-Lattice Methods

Although the HOFW method should not be classified as a vortex-lattice method
(as discussed in Section 2.2), it is most closely aligned with these methods in terms
of convergence properties due to a combination of the lack of thickness modeled as
well as the flow tangency boundary condition. According to Rusak et al. [82], there
are six parameters that may influence convergence for a vortex-lattice method:

1. the integration method for free-vortex trajectories,
2. the cutoff distance for the interaction of free vortices or a free vortex with a
   surface panel,
3. the length of the rolled-up wake,
4. the initial shedding angle of the wake,
5. the influence of the paneling scheme on the free-vortex trajectories and wing-
   body juncture, and
6. the effects of grid refinement in the wake and on solid surfaces.

The first four of these concerns are either irrelevant to this method (such as the
second item) or have been thoroughly addressed by Bramesfeld [8] for fixed-wing
cases and are not expected to vary significantly for propeller-wing systems. The
final two items on the list are of concern due to the complexity of the propeller-wing interaction and are addressed within this chapter.

With regard to these final two parameters, the authors state,

“In a finite difference solution of a differential equation, a mesh refinement should, in principle, converge to the solution of the differential equation itself, whereas in VLMs there is no reason that it should. The tangency boundary condition is satisfied at the collocation points only, while everywhere else the fluid “leaks” through the surface. This leak will not be eliminated by grid refinement. Still, the character and trends of the variation of the solution with vortex-grid refinement are indicative of the uniqueness of the solution.”

In this description, the authors delineate between methods that discretize a control volume, such as RANS and Euler solvers, and methods such as VLM and the HOFW method. For example, even though an incompressible Euler solver and a VLM exist within the potential flow domain, they will not consistently limit to the same solution with increasing resolution. Accordingly, it is not necessarily meaningful to apply methods commonly employed in CFD verification such as grid convergence indices [83].

A similarly qualitative description of the influence of grid resolution on a potential flow solution is provided by Katz and Plotkin [44]:

“It is very important to realize that the grid does have an effect on the solution. Typically a good grid selection will enable convergence to a certain solution when the density is increased (within reason). Moreover, a good grid selection usually will require some preliminary understanding of the problem’s fluid dynamics...”
In light of these descriptions, the goal of the resolution study provided in this chapter is to ensure that all major features of the lifting surfaces, wakes, and interactions are appropriately modeled, and that further refinement doesn’t appreciably reduce the uncertainty in the time-averaged solution.

4.3 Description of Representative System

4.3.1 Geometry and Operating Conditions

The representative propeller-wing system selected for this study is shown in Fig. 4.1. The system is assumed to be symmetric. The propeller is a simple twisted rectangular planform analyzed at an advance ratio \( (V_\infty/\Omega R) \) of 0.294. The wing operates at an angle of attack of 5° and the propeller axis is assumed to be parallel to the body-fixed x-axis. The free-stream velocity is 100 ft/s. In the baseline configuration, the system was run until 100 time-steps, and the last 20 time-steps were averaged.

4.3.2 Baseline Resolution

The baseline parameters for spacial and temporal resolution are provided in Table 4.1 and were chosen conservatively based on the works of Bramesfeld [8] and Basom [46].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time-step Size (( \Delta t_0 ))</td>
<td>20 time steps/rev</td>
</tr>
<tr>
<td>Wing Panels per Half-Span</td>
<td>18</td>
</tr>
<tr>
<td>Rotor Blade Spanwise Panels</td>
<td>8/blade</td>
</tr>
<tr>
<td>Chordwise Panels</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 4.1: Baseline panel and time-step resolution.
4.4 Influence of Increasing Resolution on Computational Time

The spacial resolution of the lifting surface is dictated by the number of lifting lines and the number of panels in the spanwise direction. The tradeoff for increasing resolution is an increase in computational time in both cases, as can be seen in Fig.4.2, but the source and severity of this increase differs slightly between the two.

The number of spanwise panels used to model the lifting surface has an influence on computational time because it dictates the number of control points along the span at which flow tangency is enforced, and subsequently, number of DVEs that are shed into the wake at each time-step. Consequently, the number of spanwise panels also dictates the number of control points that are convected with
local velocities to define the wake shape. Thus, as the number of panels on the lifting surface increases, the number of function calls required for wake relaxation increases as well, further extending the computational time required per time-step.

Alternatively, the number of lifting lines is only influential on the number of control points on the lifting surface. The increase in computational time due to added lifting lines is thus less severe that that due to added spanwise panels. This
comparison can be seen in Fig. 4.2 as well.

One source of computational cost that can be limited is that used to compute wake convection. After a certain distance downstream, small adjustments in wake shape no longer significantly influence the lifting surface conditions. To reduce computational time, these DVEs can remain fixed in the global reference frame with very little influence on the lifting surface conditions. A visualization of such downstream wake “freezing” is shown in Fig. 4.3.

![Visualization of wake relaxation downstream cutoff.](image)

Figure 4.3: Visualization of wake relaxation downstream cutoff.

The length of the region of the wing wake over which wake relaxation occurs, designated $l_w$, should be selected as a trade-off between the influence that the
relaxation has on the integrated time-averaged forces and the reduction in computational time. For the representative system, the influence of wake relaxation is small to begin with (particularly on the wing forces), as can be seen in Fig. 4.4. As a result, the integrated wing forces change very little with $l_w$, and thus a distance of less than a quarter chord is recommended given the significant computational savings. It is noteworthy that the wake of the propeller has already had about two chord lengths to relax prior to reaching the downstream cutoff, allowing for accurate thrust and torque predictions even in cases where the propeller is not lightly loaded.

4.5 Spatial Resolution

As previously stated, the goal of increasing the spacial resolution on the lifting surfaces is to ensure that the major features of the flow field are captured. In the case of the wing surface, these major features include both the influence of the wing wake and more visibly the influence of the upstream propeller wake. For example, two wing panel resolutions are shown in Fig. 4.5, three spanwise panels on the left and 18 spanwise panels on the right.

With three spanwise panels, there are only three control points on the wing to capture the influence of the propeller wake. This is at best a crude representation of the flow field on the surface of the wing. In addition, the interaction between the propeller wake and wing wake is very roughly captured. In contrast, with 18 wing panels, both interactions are more accurately captured.

As further proof of the necessity of increased wing panels, the convergence behavior of the lift, drag, thrust, and power coefficients of the wing and propeller with increasing number of spanwise panels is provided in Fig. 4.6. While the
thrust and power are not heavily influenced by the number of panels on the wing, both the lift and drag are not sufficiently converged with fewer than the baseline recommended resolution of 18 panels per half span.

Similarly, the number of panels used to represent the propeller blade dictates both the propeller blade surface resolution and the quality of the representation of the propeller wake, the latter of which also impacts the correctness of the lift
distribution on the wing. For example, a low resolution (four panels) and a high resolution (20 panels) solution are shown in Fig. 4.7. At the lower resolution, this discontinuity of the surface due to planar rectangular paneling is visible, along with leaks in the wake where the rough paneling is unable to accurately represent the helical wake. These issues are greatly reduced by increasing the number of panels per blade.

Varying the number of panels on the propeller influences all four measures of system performance, as shown in Fig. 4.8, although the variations in thrust and power coefficient are very small (less than 2% across the full range). This parameter has a larger effect on the forces on the wing, particularly when wake relaxation is taken into account. A visual inspection of the variation of the lift and drag with the number of propeller spanwise DVEs reveals that at least 16 is preferred for relaxed-wake analysis and at least 12 for fixed-wake analysis, representing a significant increase from the baseline recommended value. Even with increasing resolution there is variation in the relaxed wake values, resulting in a standard
deviation of ±0.32% in lift coefficient and ±0.76% in drag coefficient. Variation in
the fixed-wake values, while present, is negligible.

The reason for the increase in resolution over the baseline likely stems from
the difference in geometry and operating conditions for a propeller as compared to
a wind turbine. In particular, the spanwise twist of a propeller blade is typically
more aggressive than that of a wind turbine blade.

The number of lifting lines influences the number of control points on the lifting surface, thus improving the ability of the method to accurately represent interactions between lifting surfaces and wakes. An improvement in the method is the ability to dictate the number of lifting lines on the propeller separately from the number of lifting lines on the wing surface. This was found to be necessary due to the very different requirements for the number of lifting lines on the propeller blades versus the wing.

The number of lifting lines on the propeller blade surfaces is kept constant at one in this study for two reasons. First, the propeller blades are assumed to be of reasonably high aspect ratio (greater than four). This permits sufficiently accurate analysis with a single lifting line. Second, because of the high amount of twist in propeller blades and the approach used for geometry discretization, adding lifting lines results in increasing error due to leakage between DVEs. Although this leak is present for a single lifting line, the magnitude of the error induced by it can be
Figure 4.8: Convergence behavior of time-averaged integrated force coefficients as a function of the spanwise panel resolution on a propeller blade.

seen through comparison with experimental data.

The number of lifting lines on the wing surface changes the resolution of the interaction between the wing and propeller, as is visible in Fig. 4.9. This is particularly important for capturing the time-accurate variations in the wing forces, as is further addressed in Chapter 4. In this case, it has relatively little influence
on the time-averaged solution as can be seen in Fig. 4.10. This is likely because at the advance ratio in this case, only one wake interaction occurs on the lifting surface per spanwise location. If, for example, the advance ratio was halved, an interaction would occur twice on the surface at a given spanwise location. Thus, if only one interaction was captured, the effect would be underestimated.

Despite the small numerical influence, a few things are noteworthy in Figs. 4.9 and 4.10. First, as is visible in the Fig. 4.9, increasing the number of lifting lines shifts the first SDVE forward. As explained in Chapter 3, this is because two geometric constraints are applied when constructing a lifting surface: the trailing edge location is kept at three quarters of the length of the most aft SDVE and the total chord length of the lifting surface is kept constant. As a result, the leading edge of the first SDVE moves forward in the global reference frame with added lifting lines. This effect limits to the case where the SDVE is an exact geometrical representation of the input lifting surface. In the system, this means that the first
Figure 4.10: Convergence behavior of time-averaged integrated force coefficients as a function of the number of lifting lines on the wing.

SDVE on the wing becomes marginally closer to the propeller plane with increasing number of wing lifting lines. This effect can be seen in the very small increase in thrust and power coefficients in Fig. 4.10.

The lift and drag variation, while small in magnitude, are not convergent in nature. Still, up to 64 lifting lines, the standard deviation is $\pm 0.46\%$ in lift and
±0.91% in drag for the relaxed wake case and ±0.45% in lift and ±0.69% in drag for the fixed wake case.

4.6 Time Resolution

The size of the time-step used to model the system influences the length of the wake DVEs and thus has an impact on the accuracy of the wing wake and, more severely, the helical wake model, as can be seen in Fig. 4.11. It also dictates the rate at which the system of equations is solved for a new set of SDVE strengths. As a result, the time-step size influences both the propeller and wing performance, as shown in Fig. 4.12.

The propeller performance is well converged by 10 time-steps per revolution, whereas the lift and drag are more variable. The lift and drag seem not to be convergent in nature, but again vary a relatively small amount. The standard
Figure 4.12: Convergence behavior of integrated force coefficients as a function of the number of time steps per revolution of the propeller.

deviations in lift and drag for time-steps smaller than that required for propeller performance convergence are provided in Table 4.2.
Table 4.2: Standard deviation for lift and drag as found with the representative system for time-steps smaller than that required for propeller performance convergence.

<table>
<thead>
<tr>
<th>Wake Mode</th>
<th>Lift</th>
<th>Drag</th>
</tr>
</thead>
<tbody>
<tr>
<td>Relaxed</td>
<td>±0.55%</td>
<td>±1.37%</td>
</tr>
<tr>
<td>Fixed</td>
<td>±0.14%</td>
<td>±2.06%</td>
</tr>
</tbody>
</table>

4.7 Limitations of Convergence Analysis

Two limitations with the approach taken in this chapter should be noted. First, the interaction between resolution parameters has been neglected. For example, the integrated lift may depend both on the number of panels on each propeller blade and the number of panels on the wing surface. It was assumed that all significant effects would be visible through single variable manipulation. Second, the converged resolution parameters depend on the system being analyzed. For example, a low aspect-ratio lifting surface will likely require additional lifting lines for convergence, or a more heavily loaded propeller may require more time-steps per revolution.

4.8 Summary

The lift, drag, thrust, and torque as predicted with the DVE method for the representative propeller-wing case exhibit satisfactory behavior with increasing spacial and temporal resolution. For the case investigated, the baseline resolution parameters as provided in Table 4.1 were sufficient for capturing the major flow features with the exception of the number of panels on the propeller blade, which required double the recommended number for convergence of forces on the downstream wing surface. Uncertainty due to variation of these parameters is approximately ±2%
in drag and ±0.5% in lift and is negligible in thrust and torque.
Chapter 5

Time Dependency of Propeller-Wing Interaction

5.1 Overview

The interaction between an upstream propeller and downstream wing is time-dependent by nature. With this in mind, there are several reasons to investigate time-dependency within the HOFW method. First, the ability to model unsteady effects is a new property of the improved HOFW method and must be thoroughly explored. In addition, the time-accurate lift response for propeller-wing systems may be important for structural and aeroelastic considerations, in particular, to identify cases in which higher fidelity analysis methods are needed. Most importantly, comparison of the time-accurate response as predicted using the HOFW method with a more accepted method such as fully unsteady CFD allows for better understanding of the behavior of the HOFW model. The objective of this chapter is thus to address in detail the time dependency present in the propeller-wing interaction and the ability of the HOFW method to capture it.
To accomplish this goal, two studies are conducted. First, the approach to modeling unsteady effects is verified using the Küßner function. This study also elucidates the terms present in the unsteady response and the influence that lifting surface resolution has on these terms. Next, the time-accurate behavior of a specific propeller-wing system is discussed in detail and compared with a fully unsteady CFD analysis. These studies consider time dependency in lift only, as the approach to induced-drag calculation is only defensible when time averaged (as discussed in Chapter 3). Further investigation into time accurate drag response has been left to future work.

5.2 HOFW Modeling of a Sharp-Edged Gust Response

The variations in lift on the wing surface due to the velocities induced by the propeller are different from variations that would result from independent motion of the wing, such as pitching and plunging. In the former case, the airfoil moves at a constant velocity while the flow field accelerates, and as a result, the body-fixed coordinate system remains inertial. Alternatively, a pitching or plunging wing has an accelerating (non-inertial) body-fixed coordinate system. The delineation between non-inertial and inertial body-fixed coordinate systems is necessary because the unsteady Bernoulli equation (and transitorily, the approach to calculating lift) as derived in Chapter 3 applies only to inertial reference frames [77].

To verify that the HOFW method correctly accounts for variations in the freestream velocity (as occur in the propeller wake), a simple comparison is that of an airfoil encountering a gust. The Küßner function dictates the lift response of
a two-dimensional airfoil entering a vertical sharp-edged gust of small magnitude relative to the free-stream velocity \( \frac{v_{\text{gust}}}{V_{\infty}} << 1 \) \([38, 84, 85]\). The lift response of an airfoil penetrating such a gust is given by

\[
 c_l(t) = 2\pi \left( \frac{w_0}{V_{\infty}} \right) \psi(s),
\]

where

\[
\psi(s) = \frac{2}{\pi} \int_0^{\infty} (F(k) \cos(k) + G(k) \sin(k)) \frac{\sin(ks)}{k} dk.
\]

In Eq. 5.2, \( F(k) \) and \( G(k) \) are the real and imaginary components of the Sears function,

\[
F(k) + iG(k) = C(k) [J_0(k) - iJ_1(k)] + iJ_1(k),
\]

where \( J_0 \) and \( J_1 \) are Bessel functions. In Eq. 5.3, \( C(k) \) is the Theodorsen function, which is given by

\[
C(k) = \frac{H_1^2(k)}{[H_1^2(k) + iH_0^2(k)]},
\]

where \( H \) are Hankel functions. Because Eq. 5.2 does not have a closed form solution, it was numerically integrated for applications within this chapter.

The sharp-edged gust was modeled in the HOFW method by applying symmetry boundary conditions (zero vorticity) to the tips of a finite wing resulting in a two-dimensional analysis. The surface travels at a constant velocity in the global reference frame at a very small angle of attack (0.01°). When it reaches a specified location in the global reference frame, it experiences a step change in the “induced” velocity on the surface of about 5% of free-stream in the global
z-direction, as shown in Fig. 5.1. As is the case for all unsteady analysis within this thesis, the wake was assumed to be fixed to prevent numerical instability in the wake relaxation process due to the leading and trailing edge vortices.

Figure 5.1: HOFW model of the Küssner function.

It is first worthwhile to consider the quasi-steady response to this scenario with varying numbers of lifting lines on the lifting surface, as shown in Fig. 5.2. As would be expected, the quasi-steady response reaches the asymptotic value of lift more quickly than the fully unsteady Küssner response. Adding lifting lines damps this response primarily because it improves the control point resolution of the chordwise location of the gust at each time-step. Some additional damping occurs because the surface and wake strength iterate with time-steps, which slows the response to a change in surface conditions. For example, if the surface circulation changes, this change is shed into the wake, but the surface conditions are not updated based on this change until the next time-step. This damping is reduced by small time-step sizes such that any change in the wake is accounted for on the surface prior to it moving significantly from the lifting surface.
Figure 5.2: Quasi-steady HOFW response to a sharp-edged gust as compared with the Küssner function.

The next issue to be considered is associated with the fully unsteady response as shown with varying numbers of lifting lines in Fig. 5.3. As can be readily observed from the figure, there is a heavy dependence of the response on the number of lifting lines.
Figure 5.3: Unsteady DVE response to a sharp-edged gust as compared with the Küssner function.

To investigate the source of this dependence, it is necessary to return to the derivation of the unsteady lift equation provided in Section 3.2. Recall, the equation for lift due to the kinematic velocity of the SDVEs (Eq. 3.11) is

\[
F_k = \rho \sum_{i=1}^{m} \sum_{j=1}^{n} \left[ \int_{\eta-\eta_{i,j}}^{\eta_{i,j}} |\nabla_k \times \hat{s}| (\Gamma_{i,j} - \Gamma_{i-1,j}) d\eta 
+ \int_{-\eta_{i,j}}^{\eta_{i,j}} \frac{\partial (\xi, \eta, t)}{\partial t} (2\xi_{i,j}) d\eta \right] \left( \frac{\nabla_k \times \hat{s}}{|\nabla_k \times \hat{s}|} \right). \tag{5.5}
\]

The term that results in the seemingly erratic behavior seen in Fig. 5.3 is the \( \partial \Gamma / \partial t \) term. This term can be traced to the \( d\phi / dt \) term in the unsteady Bernoulli equation, which accounts for the non-circulatory lift, including both apparent mass
and “impulse” lift. As the lifting surface is not accelerating in this case, the apparent mass term is by definition zero, but the impulse is still present.

In the HOFW setting, the \( \partial \Gamma / \partial t \) term is a spatial integration of a time derivative of an impulse function. The time derivative is estimated by taking the difference in integrated circulation over two time-steps, but the actual change in integrated circulation occurs at a single point on the DVE (the leading edge) and is controlled by the conditions at a single point on the DVE (the control point). As a result, no matter how small the time-step is, the change in circulation is the same as long as the conditions at the control point dictate a change. Thus, \( \partial \Gamma / \partial t \) will tend towards infinity with decreasing time-step size relative to the length of the SDVE. Likewise, \( \partial \Gamma / \partial t \) will tend towards zero with increasing time-step size for the same event.

The only time-step size that makes it so that the estimated \( \partial \Gamma / \partial t \) is assuredly in the correct range is that which forces the change on the control point to have happened within the distance traversed by the control point over the last time-step. In other words, if the distance traversed by a single point on the wing over a time-step is defined by \( \Delta x_w \) and the length of an SDVE is dictated by \( \Delta x_c \),

\[
\frac{\Delta x_w}{\Delta x_c} = 1. \tag{5.6}
\]

For the time-step size used in Fig. 5.3, a single lifting line results in a ratio \( (\Delta x_w / \Delta x_c) \) of 0.33, while 12 lifting lines results in a ratio of four. The most accurate case of the three is that which uses three lifting lines results in a ratio of approximately one. In this case, it still appears that the influence is over-predicted. This is likely because the event in question is always assumed to cover the full DVE when in reality it may only cover a portion of it. This issue can be addressed by
reducing the time-step size and increasing the number of lifting lines.

As an example, the lift response when 30 lifting lines are used with an appropriate time-step size is shown in Fig. 5.4. The response closely approximates the lift response prediction using the Küssner function, with the exception of a slight discontinuity when the gust reaches the trailing edge of the lifting surface. This discontinuity is a result of the lag in wake/lifting surface strengths and will be discussed in the following paragraphs.

![Figure 5.4: High resolution (30 lifting lines) HOFW response to a sharp-edged gust as compared with the Küssner function.](image)

Also shown in Fig. 5.4 is the unsteady response when the $\partial \Gamma / \partial t$ term is omitted to characterize the relative magnitude of the unsteady terms. From this comparison it is clear that it would be inappropriate to omit either of the unsteady terms in a gust response.
On the time-step after the last control point on the surface experiences the gust, the $\partial \Gamma / \partial t$ term suddenly drops to zero on the lifting surface but the shed circulation has not yet returned to zero. This offset results in the discontinuity in the lift response. To lessen the severity of this discontinuity, a solution damping method was applied in which the $\partial \Gamma / \partial t$ calculated for the current time-step is averaged with that of the previous time-step. The impact of this approach is visible in Fig. 5.5.

![Figure 5.5: Influence of damping of the $\partial \Gamma / \partial t$ term on the lift response.](image)

The damping slightly slows the lift response prior to one semi-chord, but is otherwise very close to the original model outside of the discontinuity region. The severity of the discontinuity is greatly reduced by the damping. This becomes particularly important in the propeller-wake case, where rapid oscillations create
the potential for many large discontinuities if the solution is not damped.

5.3 HOFW Modeling of the Experiment of Dunsby et al.

The geometry used to investigate the time-dependent response of a propeller-wing system is that used in a wind-tunnel test conducted by Dusby et al. [9]. This case was selected both because the geometry heavily emphasizes propeller-wing interaction and because the experimental data provide grounding for comparison of the results. A full comparison of the integrated time-averaged force predictions for the case with the experimental data is provided in Chapter 6.

A dimensioned diagram of the model is provided in Fig. 5.6. The wing section uses an NACA 0015 airfoil, and the wing Reynolds number is approximately one million in all tests. The four-bladed propeller rotates in the down inboard orientation and has a simple rectangular planform with a linear twist distribution and NACA 1-series airfoils. The system was modeled at zero angle of attack in this chapter and a thrust coefficient of approximately 0.02 (by rotor conventions). The zero angle of attack case was selected because it heavily emphasizes the influence of the velocities induced by the propeller. The propeller was modeled according to the geometry and operating conditions described in Dunsby et al [9]. For the case considered, the propeller operated at an RPM of 3000 and the tunnel speed was approximately 95 ft/s.
There are two preliminary concerns with respect to the modeling of this system within the HOFW method. First, as is explained in Chapter 2, in unsteady analysis of propeller-wing systems, the propeller and propeller wake are assumed quasi-steady and therefore neglect shed-circulation and non-circulatory effects. In addition, the wake is assumed to be fixed. These assumptions may influence the time-accurate solutions on the wing. Second, there are two constraints on an allowable time-step size for analysis. The ratio of $\Delta x_w / \Delta x_c$ is constrained to one, as described in the previous section. The second constraint is that it is necessary to apply a time-step size such that an integer number of time-steps occur per blade passage to allow for lift sensitivity due to proximity of lifting surfaces. Because the number of lifting lines is also an integer value, this set of constraints does not necessarily allow for feasible solution. The best three solutions for the Dunsby
case, for example, are provided in Table 5.1. The percent difference between the two time-step lengths is less than 1% in all three of these cases. The integer blade passage time-step size was actually used in the proceeding cases.

<table>
<thead>
<tr>
<th>Number of time-steps per revolution</th>
<th>Number of lifting lines</th>
<th>Time-step size for unsteady effects (sec)</th>
<th>Time-step size for integer blade passage (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>13</td>
<td>0.001220226</td>
<td>0.001214726</td>
</tr>
<tr>
<td>28</td>
<td>23</td>
<td>0.000689693</td>
<td>0.000694129</td>
</tr>
<tr>
<td>44</td>
<td>36</td>
<td>0.000440637</td>
<td>0.000441719</td>
</tr>
</tbody>
</table>

Table 5.1: Possible time-step sizes to meet numerical constraints for the Dunsby case.

The paneling resolution required for this system was increased slightly from the values recommended in Chapter 4 to better resolve the time-accurate response. The wing surface was paneled with 30 panels per half span, and the propeller blade was paneled with 16 panels per blade. The time-averaged force coefficients for these three time-step sizes are shown as a function of time-step number in Fig. 5.7 for the zero alpha case along with the experimentally determined values for context. The difference between the experiment and HOFW results in lift is partially due to the fixed-wake assumption, which in this case decreases the lift coefficient by 0.02. Additional sources of error are addressed in Section 6.3.3. In terms of grid resolution, there is an increase in lift coefficient with added lifting lines and smaller time-step sizes but it is less than 0.01 in magnitude. While this is not a negligible percentage of the integrated lift coefficient in this case, it is likely attributable at least in part to the fact that the only lift developed in this case is due to interactions with the propeller and any grid sensitivity to the interaction is
thus amplified.

Figure 5.7: Variation of the time-averaged lift on the wing surface as a function of time-step number.

Also visible in Fig. 5.7 is the trade-off in terms of computational time. As the length of the time-step decreases, the number of time-steps needed for convergence increases. This issue is exacerbated by the fact that to retain the ratio of $\Delta x_w/\Delta x_c$ as the length of the time-step decreases, it is necessary to add lifting lines, increasing the computational time per time-step. As a result, this particular case becomes somewhat more computationally expensive, requiring nearly 24 hours to complete at the highest resolution. That said, the increased resolution is only necessary to resolve time-accurate unsteady forces, and is not expected to be of concern in every design case.

One final concern based on Fig. 5.7 is the difference in the time-averaged values between quasi-steady and unsteady lift coefficient. While again, this difference is very small in magnitude (less than 0.003), it is consistent across the three cases and thus worth noting.
For comparison, a fully unsteady Reynolds-Averaged Navier Stokes analysis was performed on the system using StarCCM+ from CD-Adapco [86]. This analysis required a grid of approximately 6 million cells and was run on 4 CPUs for 30 hours per case. The mesh modeled the experimental inflow, and boundary conditions applied included constant velocity inflow, constant pressure outflow, and symmetry planes for the tunnel facility walls. The wing mesh remained fixed during the analysis and interfaced with a rotating propeller mesh through a sliding interface. The solution required roughly 1000 time-steps to remove transients and an additional 100 time-steps for unsteady data. A contour plot of the pressure coefficients for the zero angle of attack case is shown in Fig. 5.8.

![Contour plot of pressure variation in the flow-field at a single moment in time as calculated using fully unsteady CFD.](image)

Figure 5.8: Contour plot of pressure variation in the flow-field at a single moment in time as calculated using fully unsteady CFD.

The time-accurate lift response with varying numbers of lifting lines is provided in Fig. 5.9. The qualitative appearance of the quasi-steady oscillation changes with increased resolution and even at the highest resolution does not match the smooth
curve calculated using CFD. Despite this, the frequency content is fairly static with a main peak at 200 hz (1 per blade passage), as is visible in Fig. 5.10. The magnitude of this peak matches that of the CFD response at the lowest resolution (13 lifting lines), but this results in significant over-prediction once unsteady effects are included (as can be seen in Fig. 5.11). A comparison of the magnitude of the 200 hz peak for each case is provided in Table 5.2.

Figure 5.9: Comparison of the time-accurate lift predictions with CFD as with varying numbers of lifting lines and time-step sizes.

There is also over-amplification in both the quasi-steady and unsteady HOFW
Figure 5.10: Comparison of the frequency response of quasi-steady lift predictions with varying numbers of lifting lines and time-step sizes.

<table>
<thead>
<tr>
<th>Lifting Lines</th>
<th>Quasi-Steady</th>
<th>Unsteady</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>0.0565</td>
<td>0.1157</td>
</tr>
<tr>
<td>23</td>
<td>0.0368</td>
<td>0.1842</td>
</tr>
<tr>
<td>36</td>
<td>0.0232</td>
<td>0.0377</td>
</tr>
<tr>
<td>CFD</td>
<td>0.0567</td>
<td>0.0567</td>
</tr>
</tbody>
</table>

Table 5.2: Amplitude of peaks at 200 hz for different resolutions and modes of analysis.

response of the much smaller peaks present in the CFD response at 50 hz (1 per propeller revolution), 400 hertz, and 600 hertz (2 per and 3 per blade passage respectively). The latter is missing in the lowest resolution response (13 lifting lines), as it can only be resolved to 400 hz as constrained by the Nyquist frequency due to the sampling rate.

The unsteady response for the middle resolution (23 lifting lines) seems partic-
Figure 5.11: Comparison of the frequency response of unsteady lift predictions with varying numbers of lifting lines and time-step sizes.

ularly volatile, with significant overreactions at all relevant frequencies. Because the quasi-steady response is well behaved, it is apparent that the problem lies in either the shed-circulation term or the $\partial \Gamma / \partial t$ term. As the former tends to damp the response, the most likely culprit is the $\partial \Gamma / \partial t$. In agreement with this hypothesis, the percent error in time-step size for this case is the largest of the three (see Table 5.1), which may contribute to the response. It is additionally possible based on the Küssner study that an accurate response prediction requires more lifting lines. The well-behaved 36 lifting line response would seem to support this theory.

A few conclusions can be drawn with regards to the lift response. First, the frequency of the main peak as predicted by CFD and the HOFW method aligns with one per blade passage. This is intuitive, but necessary to verify (particularly
in terms of the HOFW response) given the complexity of the wake interaction. Second, the time-accurate response requires very high resolution when compared to that needed for decent time-averaged response, and is still lacking in some respects. This reaffirms that this approach should be used sparingly in design, and primarily to identify potential issues to be addressed with higher-fidelity methods such as unsteady CFD. Finally, the magnitude of the oscillation response is small (less than 0.1 lbs or about 0.01 in lift coefficient) and although this case represents a fairly lightly loaded propeller, it does heavily emphasize the propeller-wing interaction. This result, while intuitive, was necessary to verify through analysis.

At angles of attack larger in magnitude, the time-accurate response predicted by the HOFW method becomes less coherent due to the high curvature in the propeller wake. Still, the amplitude of the 200 hz response in percent of the time-averaged lift coefficient is decently represented in magnitude and trend, as can be seen in Table 5.3. In these cases, the magnitude of the unsteady CFD response is larger than the quasi-steady response and smaller than the fully unsteady response.

<table>
<thead>
<tr>
<th>$\alpha$</th>
<th>CFD</th>
<th>HOFW Quasi-Steady</th>
<th>HOFW Unsteady</th>
</tr>
</thead>
<tbody>
<tr>
<td>$-5^\circ$</td>
<td>2.09%</td>
<td>1.66%</td>
<td>4.11%</td>
</tr>
<tr>
<td>$-10^\circ$</td>
<td>1.52%</td>
<td>0.87%</td>
<td>1.83%</td>
</tr>
</tbody>
</table>

Table 5.3: Comparison of the CFD and HOFW (modeled with 36 lifting lines) predicted amplitude of the 200 hz peak response as a percent of the time-averaged lift coefficient.

5.4 Summary

There are two effects accounted for within fully unsteady analysis that are not included in quasi-steady analysis. The first is the shed circulation, which serves to damp the response. The second, and more volatile of the two, is the $\partial \Gamma / \partial t$. 
In order to accurately represent this quantity it is necessary to keep a ratio of the length on the SDVE in question to the distance travelled by the SDVE to one. With this constraint applied, the HOFW method is capable of recreating the Küssner function response to a sharp-edged gust accurately at high resolution (large number of lifting lines and small time-steps). To account for a small discontinuity that develops when the gust reaches the trailing edge, the $\partial \Gamma / \partial t$ term is damped through averaging with the previous time-step’s solution.

The complexity of the propeller-wing interaction provides further challenges. First, the time-step must be set such that there are an integer number of time-steps between blade passages. This is a result of the fact that the lift response is sensitive to the proximity of the propeller blade to the wing. This additional constraint results in a potentially infeasible solution to the question of time-step size. To allow for a solution, very slight relaxation of the ratio constraint adds potential error to the $\partial \Gamma / \partial t$ term. The resulting unsteady lift response is similar to that found in CFD in magnitude and frequency at high resolution.
Chapter 6

Comparison with Experimental Results and Other Analytical Methods

6.1 Overview

The objective of this section is to provide evidence for the validity of the HOFW method through comparison with both analytical results using other methods and experimental results. Because the method has been previously examined for fixed-wing analysis, the focus of this chapter is propellers/proprotors and propeller-wing systems. The first section investigates two propulsive systems - a traditional general aviation propeller and a proprotor. Next, propeller-wing systems are investigated, beginning with a comparison of the HOFW method with the semi-empirical method of Smelt and Davies [13] and the jet-flap approach recommended by McCormick [14]. The experiment of Dunsby et al. [9] is then used for comparison of time-averaged values as calculated by both the HOFW method and fully un-
steady CFD. The latter is additionally helpful for providing details of the spanwise lift and drag distribution that are not available in the experimental results. Finally, the HOFW method was used to predict the trends found experimentally by Veldhuis [3].

6.2 Propeller/Proprotor Performance

6.2.1 P5868-9 Propeller

Propeller 5868-9 was a 10 ft diameter propeller with Clark-Y airfoil sections tested by the National Advisory Committee for Aeronautics [10]. The chord, pitch distribution, and thickness properties of the propeller are provided in Fig. 6.1. The pitch distribution provided is based on the mean chord line of the sections. It was converted to a geometric pitch angle distribution which was then adjusted to account for the zero-lift line of each section via a simple correction recommended by McCormick [25]. Three average thicknesses were used for the prediction of sectional profile drag.
Figure 6.1: The geometry of propeller 5868-9 in terms of non-dimensional section thickness (t/c), non-dimensional chord distribution (c/D), and pitch distribution (p/D) [10].

The predicted power coefficient (according to propeller convention), $C_P = \frac{P}{(\rho n^3 D^5)}$, and propulsive efficiency, $\eta$, as a function of advance ratio for three blade pitch angle settings are shown in Figs. 6.2 and 6.3, respectively. Across all three settings, fixed- and relaxed-wake results are very similar to one another, with slight departures as the advance ratio approaches zero. This trend is indicative of the influence of the wake shape at low advance ratios. The fixed-wake prediction assumes an axial induction of zero, an assumption that breaks down as the advance ratio approaches zero.
Figure 6.2: A comparison of the power coefficients of the P5868-9 propeller as a function of advance ratio as predicted using the HOFW method and experimentally derived [10].
Figure 6.3: A comparison of the propulsive efficiency of the P5868-9 propeller as a function of advance ratio as predicted using the HOFW method and experimentally derived [10].

In lightly loaded conditions (moderate to high advance ratios and low to moderate pitch angles), the predictions match the experiment well. At 15° pitch, the average error in efficiency across the full range of advance ratios is approximately 2%, increasing to 4% at 25° pitch and 10% at 35° pitch. The errors across all pitch angles are driven primarily by departures at the low end of the range of advance ratios. In this region, errors are attributable to shortcomings in the stall model, which is simplistic and intended only for the identification of stall onset rather than post-stall performance prediction. A second source of error lies in the implicit assumption that the section lift-curve slope is a consistent $2\pi$ per radian, when in reality the lift-curve slope depends on the airfoil and decreases significantly as the
airfoil approaches stall. As the propeller pitch angle increases, it is possible that both of these errors come into effect, resulting in the increased error at increased pitch. It is additionally possible that geometric discrepancies between the test propeller and modeled propeller are a contributing factor. As a result, the current method is best suited for lightly loaded propellers.

6.2.2 JVX Proprotor

The JVX proprotor [11, 12, 87] is frequently referred to as a 0.656-scale V-22 proprotor, although in reality there are slight differences between the two designs. It provides a good basis for validation of the method for proprotors. The geometry of the JVX is described in Fig. 6.4, and the discretized geometric parameters as modeled within the HOFW method are shown in Table 6.1. The first panel defined is discretized into seven elements, the second into three elements, and the third into ten elements.
Figure 6.4: Geometry of JVX Proprotor in terms of non-dimensional section thickness (t/c), non-dimensional chord distribution (c/R), and section twist in degrees [11].

Table 6.1: Table of geometric parameters used to model the JVX Proprotor.

<table>
<thead>
<tr>
<th>Panel Number</th>
<th>y_L (ft)</th>
<th>y_R (ft)</th>
<th>c_L (ft)</th>
<th>c_R (ft)</th>
<th>(\beta_L) (deg)</th>
<th>(\beta_R) (deg)</th>
<th>(\frac{L}{c})</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.125</td>
<td>5.938</td>
<td>1.954</td>
<td>1.681</td>
<td>36.430</td>
<td>10.000</td>
<td>28</td>
</tr>
<tr>
<td>2</td>
<td>5.938</td>
<td>7.688</td>
<td>1.681</td>
<td>1.584</td>
<td>10.000</td>
<td>4.600</td>
<td>18</td>
</tr>
<tr>
<td>3</td>
<td>7.688</td>
<td>12.5</td>
<td>1.584</td>
<td>1.310</td>
<td>4.600</td>
<td>-5.185</td>
<td>11</td>
</tr>
</tbody>
</table>

The JVX proprotor was investigated with the HOFW method over a sweep of collectives at two advance ratios resulting in the performance predictions shown in Fig. 6.5. Only the fixed-wake results are shown, as no visible difference between
fixed- and relaxed-wake results was found. Also shown in Fig. 6.5 is a performance prediction using CAMRAD II [12] and the original test data [12]. In Fig. 6.5, the advance ratio and coefficients are defined according to the rotorcraft conventions ($\mu = V_\infty / \Omega R$, $C_P = P/(\rho AV_{tip}^3)$, and $C_T = T/(\rho AV_{tip}^2)$), and $\sigma$ refers to the rotor solidity (ratio of blade area to disk area). The performance predicted by the HOFW method (FreeWake) closely matches the test data and the predictions using CAMRAD II both in shape and magnitude. This would be expected based on the P5868-9 results, as the proprotor is lightly loaded in the cruise condition.

Figure 6.5: A comparison of the predicted via the HOFW method and CAMRAD II [12], and experimentally derived [12] JVX Proprotor performance.
6.3 Propeller-Wing Systems

6.3.1 Semi-Empirical Model of Smelt and Davies

In 1937, Smelt and Davies [13] published a study that described a semi-empirical method for calculating the increase in lift coefficient of a clean (non-flapped) wing due to a lightly-loaded, upstream propeller. This model was then validated with wind-tunnel and flight test results. According to this model (following the notation of McCormick [14]), the change in lift of a wing due to a propeller slipstream can be estimated as

\[
\Delta C_L = c_l \frac{D_1 c_w}{S} \frac{w_1}{V} \lambda
\]

(6.1)

where \(D_1\) is the diameter of the propeller, \(c\) is the chord of the wing, \(w_1\) is the axial induced velocity, \(S\) is the wing area and \(V\) is the freestream velocity. The lift coefficient, \(c_l\), is the average two-dimensional lift coefficient of the region of the isolated wing that would be engulfed in the propeller slipstream. The lift factor, \(\lambda\), lies between one and two with limiting cases derived by assuming that the wing chord is much larger than the slipstream diameter (for \(\lambda = 1\)), and much smaller than the slipstream diameter (for \(\lambda = 2\)). In the latter case, it was assumed that the induced effects due to the added vorticity on the wing are negligible, which is akin to assuming that wing can be treated as practically two-dimensional. The lower bound is explained as the case in which shed vorticity is dominant, although the explanation is qualitative in nature and does not directly justify the quantitative value [88].

To compare results of this theory with performance predictions using the HOFW
method, a series of wings of constant span \( b = 10 \text{ft} \) and varying chords were investigated at an angle of attack of \( 5^\circ \) in the wake of two propellers designed for lightly-loaded conditions \( (C_T \approx 0.2 \text{ and } C_T \approx 0.8) \) using QMIL [1]. The wings were each analyzed in isolation to determine \( c_l \) for use with Eq. 6.1. The results of this study are provided in Fig. 6.6 and Fig. 6.7.

![Graph showing comparison of lift factor, \( \lambda \), calculated using the HOFW method and that suggested by Smelt and Davies [13]](image)

Figure 6.6: A comparison of the lift factor, \( \lambda \), calculated using the HOFW method and that suggested by Smelt and Davies [13]
The lift factor calculated with the HOFW method (as shown in Fig. 6.6), exhibits the correct increasing trend with increasing aspect ratio for both values of thrust. It appears the HOFW method predicts a higher low aspect ratio bound, or alternatively, the aspect ratio must be much lower than one to reach the Smelt and Davies lower bound. This result is in agreement with Obert’s [88] concerns about the lower bound, although further studies would be needed to verify.

The change in lift coefficient (as shown in Fig. 6.7) gives a more clear picture of the agreement between the methods. The general magnitude and trend of the increase in lift with the aspect ratio of the submerged portion of the wing are in agreement. While the percent difference of the $\Delta C_L$ estimate can be somewhat
high (an average of 16% over all cases), the difference in $C_L$ predicted is much smaller (an average of 1% over all cases).

### 6.3.2 McCormick’s Jet-Flap Approach

McCormick [14] recommends treating the section of the wing submerged in the propeller slipstream with the semi-empirical method developed for analysis of jet flaps. The angles involved in this analysis are defined in Fig. 6.8. In this figure, $\alpha_p$ is the propeller angle of attack, $\alpha_s$ is the slipstream angle of attack, $\alpha$ is the wing angle of attack, $\delta$ is the flap angle, and $\theta$ is the angle through which the slipstream is turned.

![Diagram of jet flap angles](image)

**Figure 6.8: Definition of angles and lengths relevant to jet flap analysis [14](reprinted with permission).**

The lift coefficient for the propeller-wing combination according to this ap-
The approach is given by

\[ C_L = C_{L,T=0} + C_{L,\Gamma} + C_\mu \sin \alpha_s + \theta \left[ \frac{\sin \alpha_p}{\sin \alpha_s} - \left(1 - \frac{F}{T} \right) \frac{V_R^2}{4V'w} \right] , \quad (6.2) \]

where \( C_{L,T=0} \) is the lift coefficient of the isolated wing, \( C_{L,\Gamma} \) is the increase in circulation due to the propeller slipstream, and the last term accounts for the turn of momentum in the slipstream. The momentum coefficient, \( C_\mu \), for the jet-flap equivalent of a propeller slipstream is given by

\[ C_\mu = \frac{NT}{qS} \quad (6.3) \]

where \( N \) is the number of propellers, \( T \) is the thrust per propeller, \( q \) is the free-stream dynamic pressure and \( S \) is the projected wing area immersed in the slipstream. The thrust recovery factor, \( \frac{F}{T} \), is a function of the type of flap and \( \theta \) and was determined for this study by using a look-up table of digitized data from McCormick [14]. The angle, \( \theta \), likewise, is provided graphically as a function of the ratio of the flap chord to the propeller diameter, and was determined similarly. \( V_R \) is the net resultant velocity consisting of the influence of the freestream and the slipstream axial induced velocity. \( V' \) is given by

\[ V' = \left[ (V + w \cos \alpha_p)^2 + (w \sin \alpha_p)^2 \right]^{\frac{1}{2}} . \quad (6.4) \]

Finally, the increase in circulation due to the propeller slipstream \( (C_{L,\Gamma}) \) is the integrated effect over a two-dimensional jet flap, \( (C_{l,\Gamma}) \) as given by

\[ C_{l,\Gamma} = C_{l,\alpha} \alpha + C_{l,\delta} \delta \quad (6.5) \]
where both coefficients are given as a function of $C_{m}u$, viz.

$$C_{l,\alpha} = 1.152\sqrt{C_{\mu}} + 0.106C_{\mu} + 0.051C_{\mu}^{\frac{3}{2}}$$ \hspace{1cm} (6.6)$$

and

$$C_{l,\delta} = 3.54\sqrt{C_{\mu}} - 0.675C_{\mu} + 0.156C_{\mu}^{\frac{3}{2}},$$ \hspace{1cm} (6.7)$$
both with units of 1 per radian.

For a simple comparison, the same set of wings of varying aspect ratio used in the Smelt and Davies study was analyzed with the approach recommended by McCormick. The results of this study are provided in Fig. 6.9.

Figure 6.9: A comparison of the change in lift coefficient due to the propeller slipstream as calculated using the HOFW method, the method recommended by Smelt and Davies [13], and the jet-flap method recommended by McCormick [14]
The three methods seem to agree on the general magnitude of the increase in $C_L$ and, at least for the higher thrust coefficient, the trend as well. Without experimental data with which to compare, it is difficult to determine which of the three models is “best” for this application. If the average of the two semi-empirical methods is used as the true value, the average error in the HOFW prediction of the more lightly loaded case is around 6% and jumps to around 13% for the more heavily loaded case. Overall, the agreement provides confidence in the ability of the HOFW method to predict trends for a simple propeller-wing system, which is vital if the method is to be trusted to predict the performance of more complex systems.

6.3.3 Comparison of Performance Predictions and Experimental Results of Dunsby et al.

The forthcoming comparison is based on the experiments conducted by Dunsby et al. [9]. The details of the geometry and operating conditions are provided in Section 5.3, along with the details of the unsteady CFD analysis.

The results of the HOFW analysis here differ from those found in Section 5.3 in several ways. First, all results shown here are conducted at a time-step size such that a full revolution of the propeller occurs over 20 time-steps. In contrast, the time-accurate solutions shown in Section 5.3 used time-step sizes that varied based on the number of lifting lines to account for the unsteady constraints described in Section 5.2. Second, all results shown within this section are time-averaged, whereas the results in Section 5.3 primarily represent time-accurate results. Third, the analysis shown here was conducted with wake relaxation, which was omitted from the results in Section 5.3 due to the instability that results from unsteady
analysis with wake relaxation. Finally, all results shown in this section were conducted with quasi-steady analysis in following with the conclusions of Chapter 5.

The time-averaged integrated lift coefficient as a function of angle of attack is shown in Fig. 6.10. The CFD best matches the experimental results across the range of angles of attack with an average difference of 0.035 in lift coefficient. The HOFW method, with an average difference of 0.040 in lift coefficient, is only slightly less accurate. Smelt and Davies method is fairly close despite the simplicity of the model, with an average difference of 0.054 in lift coefficient, while the jet-flap approach recommended by McCormick is less successful for this geometry (particularly in the high and low angles), with an average difference of 0.213.
Figure 6.10: A comparison of the lift coefficient calculated as a function of angle of attack for the Dunsby experiment [9] as calculated using unsteady CFD, the HOFW method, the method recommended by Smelt and Davies [13], and the jet-flap method recommended by McCormick [14].

To get a more detailed view of what is happening in terms of the lift, a comparison of the lift distribution at zero angle of attack as found experimentally and compared with the HOFW method and CFD method is provided in Fig. 6.11. The methods of Smelt and Davies and McCormick are not capable of predicting this information. While the influence of the propeller seems to be somewhat over-predicted in both HOFW and CFD, the trend is well represented. In addition, the matching between the CFD and HOFW is remarkable given the difference in complexity of the models.
Figure 6.11: A comparison of the spanwise lift distribution for the Dunsby experiment [9] at zero angle of attack as calculated using unsteady CFD and the HOFW method.

The time-averaged integrated drag coefficient as a function of angle of attack is shown in Fig. 6.12. Only the CFD and HOFW results are shown here because Smelt and Davies and the McCormick method do not explicitly address drag.

The error in between the models and experiment is much higher in drag than was seen in the lift response. This error can be attributed to several factors. First, the experimental results are based on force balance measurements where the measured thrust is subtracted to estimate drag. The authors of the study present doubts with respect to the quality of the thrust measurement, citing measurement accuracy within 5%. Because the magnitude of the drag is small compared to the thrust, this uncertainty translates into concerns about the drag estimation as well.
Second, the CFD and HOFW models do not account for nacelle effects, which become increasingly dominant due to pressure drag at $\pm 10^\circ$.

![Figure 6.12: A comparison of the drag coefficient calculated as a function of angle of attack for the Dunsby experiment [9] as calculated using unsteady CFD and the HOFW method.](image)

The trend in drag coefficient is somewhat similar between the two methods, but there is a difference between the models and experiment from $-5^\circ$ to $+5^\circ$ in that the increase in drag seen in the experiment is skewed to lower angles of attack, whereas the model increase is skewed to higher angles of attack in both cases. This could be a function of the influence of the nacelle or error in the experimental results, but the source of the discrepancy is unclear.

There is an average difference of 0.02 in drag coefficient between the CFD and HOFW method across the angles of attack, where the HOFW method consistently
under-predicts relative to the CFD. To investigate this discrepancy further, the spanwise distribution of drag at zero angle of attack is provided in Fig. 6.13. Given the remarkable consistency in trends across the span, particularly with regard to the influence of the propeller wake, the difference between the two can be attributed to a combination of an under-prediction of the magnitude of the profile drag component determined through look-up tables and the a significant difference in thrust and torque predicted, as will be addressed next. Increased thrust, in particular, suggests an increase in dynamic pressure within the slipstream that would increase the local profile drag.

![Figure 6.13: A comparison of the spanwise drag distribution for the Dunsby experiment [9] at zero angle of attack as calculated using unsteady CFD and the HOFW method.](image)

Both semi-empirical approaches for wing lift require propeller thrust as an
input, which presents an additional challenge with respect to use of these methods. In the case of the Dunsby experiment, the thrust was determined experimentally (albeit with the caveats mentioned previously), and could thus be implemented directly. If the thrust is not known a priori for a given system under a specific operating conditions, it must be estimated or calculated using a different analysis method. The other method may or may not allow for the influence of the wing as well, resulting in the possible need for a third correction. In the HOFW method, the thrust is predicted using the same construct as the lift and drag, and thus, changes in operating conditions and the influence of the wing are automatically taken into account. The thrust predicted by the HOFW method and CFD as compared with the experimental values are provided in Fig. 6.14.

Figure 6.14: A comparison of the thrust as a function of angle of attack as measured in the Dunsby experiment [9] and with the HOFW method and CFD.
A few observations should be noted from the results in Fig. 6.14. First, the CFD and HOFW thrust coefficient predictions differ in magnitude by 0.04 or approximately 25%. The source of this discrepancy is unclear, but it is potentially the cause of the increased profile drag seen in Fig. 6.13. Second, the trend in thrust with angle of attack present in the CFD results matches that present in the HOFW results in both shape and magnitude. This trend is also in agreement with the experimental results at low angles of attack (−5° to +5°). At the higher angles of attack (±10°), the experimental thrust increases more drastically than is predicted by either analysis method. It is plausible that this departure is cause by the lack of accounting for nacelle effects in the analysis methods and/or by errors within the experiment. The latter conclusion is anecdotally supported by experimental data from Kuhn and Draper [80] that indicates that an increase in thrust of 20% from the zero alpha condition for a propeller-wing system would require an increase in angle of attack of approximately 30°. Finally, the magnitude of the HOFW predictions are relatively close (within 5%) of the experimental results in the low angle of attack range.

In addition to thrust, the HOFW is capable of predicting the torque of the propeller to provide a full prediction of the performance of the system. This again is not covered by the semi-empirical methods directly and would require the addition of a secondary model or approach. A comparison of the torque coefficient \( C_Q = Q/\rho n^2 D^5 \) as predicted by the HOFW method with experimentally derived values are provided in Fig. 6.15. The results are as would be expected, given those shown in Fig. 6.14. The trend agreement between the CFD results and HOFW results remains present, as well as the significant difference in average magnitude. Finally, the magnitude of the HOFW prediction of torque remains relatively close
to the experimental results at low angles of attack.

Figure 6.15: A comparison of the torque as a function of angle of attack as calculated in the Dunsby experiment [9] and with the HOFW method and CFD.

Overall the HOFW method satisfactorily predicts the thrust, torque, and lift of the propeller-wing system examined by Dunsby et al. It is more accurate in lift than either semi-empirical method, and provides much more detailed information (such as lift and drag distributions) than would be readily available from these methods. Confidence in the approach taken to force predictions within the HOFW method is increased through favorable comparison with CFD in spanwise lift and drag distributions.
6.3.4 Comparison of Performance Predictions and Experimental Results of Veldhuis

As discussed in Chapter 1, the most important definition of accuracy with respect to the application of the HOFW method to design is not that the magnitude of the performance prediction for a specific design matches that determined experimentally. Rather, it is much more important that the method predict trends in performance with changes in design accurately. In order to demonstrate the ability of the HOFW method to correctly identify those trends, the experimental results of Veldhuis [3] were employed.

Veldhuis [3] conducted a series of experiments to determine the influence of the position of an upstream propeller on wing performance characteristics. To do so, he used a model in which the propeller was separated from the wing such that its position could be varied independently. A photo of the experimental setup along with the geometry of the model are provided in Fig. 6.16.
Figure 6.16: Photo of the experimental setup and geometry (in mm) of the model used by Veldhuis to investigate the influence of propeller position on downstream wing performance [3] (reprinted with permission).

Although specific propeller geometry was not provided in the reference, the thrust coefficient was provided for each case. As explained in Section 1.3.1.1, matching thrust coefficient does not necessitate an identical distribution of in-
duced velocities, and is thus not sufficient to ensure exact similarity between the HOFW model and the experiment. Still, assuming that the variations are of limited magnitude, the trends can be expected to be similar.

The first study modeled within the HOFW method is that of the influence of propeller spanwise location on wing performance at a set angle of attack (in this case, 4.2°). The lift-to-drag ratio (L/D) found using the HOFW on this series of geometries as compared to those found experimentally by Veldhuis are provided in Fig. 6.17. In this study, the propeller axis location was varied from approximately 30% of the half-span to the wing tip with the rotation direction being up-inboard. The propeller vertical axis location was kept constant in the same plane as the wing and the study was conducted at a low thrust coefficient \((C_T = 0.025)\).
Figure 6.17: A comparison of the lift-to-drag ratio of a propeller-wing system at a constant angle of attack with varying spanwise location of the propeller as calculated using the HOFW method and as determined experimentally by Veldhuis [3]. The propellers used in the two cases were different, presumably resulting in the difference in magnitude between the trends.

There is agreement between the experiment and HOFW model in trend, with increasing L/D as the propeller moves outboard. The magnitude of the L/D overall and the change in trend of the L/D is smaller than that seen in experiment. This difference can be attributed at least in part to differences in the geometry of the propeller. This trend could not be predicted with either the method of Smelt and Davies or the jet-flap approach previously discussed as neither account for drag in any sophisticated manner.

The second study reproduced is that of the influence of the direction of rotation of the propeller as a function of lift coefficient. The difference in drag between
up-inboard and up-outboard rotation directions as a function of $C_L^2$ is shown in Fig. 6.18 for a thrust coefficient of 0.168. This study was conducted with a second model of identical dimensions as those shown in Fig. 6.16 but in which the propeller was integrated into the wing with a nacelle at a set spanwise location ($y_p/(b/2) = 0.469$). In this study, the direction of rotation was varied along with the angle of attack of the model.

![Figure 6.18](image-url)

Figure 6.18: A comparison of the difference in drag due to a switch from up-outboard to up-inboard rotation orientation as a function of the lift coefficient as calculated using the HOFW method and as determined experimentally by Veldhuis [3]. The propellers used in the two cases were different, presumably resulting in the difference in magnitude between the trends.

Once again, the trend present in the experimental data matches that seen in the HOFW method, while the magnitude of the difference is exaggerated in the HOFW method. The difference in magnitude can be attributed to differences in
propeller geometry. These trends could not be captured by the method of Smelt and Davies, nor the jet-flap approach, as neither method has the ability to account for rotation direction in lift, let alone in drag.

6.4 Summary

In this section, the HOFW method was shown to adequately predict propeller and proprotor performance outside of the range of stall conditions. In addition, the method compared favorably with the method of Smelt and Davies and the jet-flap approach recommended by McCormick for performance prediction of simple propeller-wing systems. The method was then used to predict the performance of a propeller-wing system in which the propeller diameter was on the same order of magnitude as the wing half span. The results of this calculation were then compared with test results, as well as fully unsteady CFD and the low-order methods. The HOFW method was second only to the more computationally expensive CFD method in terms of the accuracy of the predictions. In addition, confidence in the method was provided through comparison of the spanwise distribution of wing lift and drag with experimental results and CFD results. The trends visible in the results matched favorably in both lift and drag, with similar magnitudes of variation due to the propeller. The results of an experimental study conducted by Veldhuis concerning the performance of the wing as a function of propeller rotation direction and spanwise location were also reproduced albeit with a different propeller than was used in the initial study. The trends predicted by the HOFW method matched those found in the experiment. In all, the method was shown to be capable of capturing the magnitude and trends in performance of the propeller-wing systems discussed as well as, if not better than, the alternative analysis options
considered.
Example Applications

7.1 Overview

Two design studies were conducted to prove the utility of the HOFW method in the context of design. The first study investigates the twist distribution on the wing for a tiltrotor with the goal of minimizing the wing drag. This study serves mainly to highlight the types of analysis enabled by the HOFW method in terms of its ability quickly and successfully explore the design space of a propeller-wing system. The second study is an investigation of the influence of rotation direction on drag at a cruise condition for a distributed electric propulsion aircraft.
7.2 Twist Optimization of Large Civil Tiltrotor Wing

7.2.1 Background on the Large Civil Tiltrotor

A series of studies was conducted within NASA’s vehicle systems program from 2001-2004 on a class of vehicles referred to as “runway independent aircraft” or RIA. The goal of this aircraft class was to reduce runway and terminal area congestion through use of VTOL vehicles for short flights. Estimates indicated that RIA could accommodate 10% of flights, allowing for a 79% reduction in airport delays. The initial mission profile determined for an RIA consisted of carrying 120 passengers over a range of 1200 nm at a cruise speed of 350 knots [89].

One outcome of this study was the Large Civil Tiltrotor (LCTR) vehicle. The most recent iteration is shown in Fig. 7.1, the LCTR2-02. The design specifications for this aircraft along with its predecessor, the LCTR1, are compared in Table 7.1.
Figure 7.1: Three view of the LCTR2-02 [15].
<table>
<thead>
<tr>
<th>Parameter</th>
<th>LCTR1</th>
<th>LCTR2-02</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gross weight (lbs)</td>
<td>123,562</td>
<td>103,600</td>
</tr>
<tr>
<td>Rotor diameter (ft)</td>
<td>88.7</td>
<td>65</td>
</tr>
<tr>
<td>Disk loading (lbs/ft²)</td>
<td>10</td>
<td>15.6</td>
</tr>
<tr>
<td>Hover tip speed (ft/s)</td>
<td>650</td>
<td>650</td>
</tr>
<tr>
<td>Cruise tip speed (ft/s)</td>
<td>350</td>
<td>350</td>
</tr>
<tr>
<td>Number of blades per rotor</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>Drag (D/q in ft²)</td>
<td>37.3</td>
<td>34.6</td>
</tr>
<tr>
<td>Wing loading (lbs/ft²)</td>
<td>80</td>
<td>107.4</td>
</tr>
<tr>
<td>Range (nm)</td>
<td>1,200</td>
<td>1,000</td>
</tr>
<tr>
<td>Cruise altitude (ft)</td>
<td>30,000</td>
<td>28,000</td>
</tr>
<tr>
<td>Cruise speed (kts)</td>
<td>350</td>
<td>300</td>
</tr>
<tr>
<td>Cruise L/D</td>
<td>11.1</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 7.1: Evolution of the design specifications of the LCTR. [15,89]

The objective of this study is to minimize the drag for the cruise condition of the LCTR2. This objective is interesting for a few reasons - first, because an approximate inviscid solution can be found based on the work of Munk [37] for comparison, and second, because it is undoubtedly influenced by the interference between the propeller and the wing and thus provides a good example of design space exploration within this context.
7.2.2 Inviscid Iterative Twist Optimization of Planar and Non-planar Wing Concepts

Two wing design cases were considered in depth with inviscid analysis - a planar wing, and a wing with a non-planar tip having a height (defined as the perpendicular distance from the wingtip to the plane of the wing) of 10% of the span, and 70° of dihedral. The second wing was intended to mimic a wing with a winglet while keeping the projected wing span and area for both cases the same, and the same as those of the LCTR2. The resulting geometries are depicted in Fig. 7.2. The analysis was completed at the cruise thrust and lift condition ($C_L \approx 0.78$) and without viscous effects to isolate the vortex-induced drag contribution.

Figure 7.2: Baseline geometry for a new planar wing (top) and wing with 10% winglet (bottom) for the LCTR2.

In this study, the wing was iteratively twisted to match the ideal loading condition. This process was completed in two main steps. First, it was necessary to identify the ideal loading condition. The details of the process used to identify
this condition are provided in Appendix E. To complete this process, the velocity distribution within a converged proprotor slipstream is needed. The proprotor was therefore analyzed without the wing and the velocity distribution was extracted. The second step of the process was to iteratively modify the wing to achieve the ideal loading condition. The simple approach taken for this modification was to twist the wing as dictated by the difference between the desired ideal loading and actual loading on each panel divided by the lift curve slope, and reanalyze the resulting geometry. This process was repeated for between five and ten iterations for each baseline, with convergence indicated by the minimum drag achieved.

The ideal lift distributions calculated for the planar wing case both with and without proprotor effects are shown in Fig. 7.3. The influence of the proprotor is apparent in the increased inboard and decreased outboard loadings.
Figure 7.3: The ideal loading of the LCTR2 planar wing with and without the proprotor.

The twist distributions found to achieve the ideal loading are shown in Fig. 7.4. The twist is defined as the zero-lift line angle as offset from an arbitrarily selected angle of attack of eight degrees, with positive twist defined in the nose-up direction. Towards the wingtip the panel resolution is insufficient to model the vertical gradient of the ideal lift distribution and was thus held constant outboard of approximately 95% span. These results show that not only does the ideal loading change when proprotor effects are included, but that the design necessary to achieve that loading changes as well. In other words, a wing twisted to achieve the elliptical lift distribution without proprotor induced velocities will not achieve the ideal lift distribution automatically once the proprotor is added.
Figure 7.4: Twist distribution for ideal loading of the LCTR2 planar wing.

The induced drag coefficients calculated for each configuration are provided in Table 7.2. Twisting the wing for ideal loading without the proprotor (i.e. the elliptical lift distribution), reduced the induced drag by about 1%, indicating that the wing is fairly efficient in terms of planform without proprotor effects. Adding the proprotor to the untwisted wing reduces the induced drag by nearly 24%. From a systems perspective, the drag reduction likely comes at a cost in terms of an increase in proprotor torque from an isolated proprotor condition. This effect was not investigated in this study but does feature in the study in Section 7.3. A reduction in induced drag would be expected, as nearly the whole wing operates in upward induced flow from the proprotor. An additional 2% reduction in induced drag can be achieved through twisting for ideal loading.
Table 7.2: Comparison of vortex-induced drag coefficients for an untwisted and twisted LCTR2 planar wing.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>$C_{D,i}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Untwisted, no proprotor</td>
<td>0.0174</td>
</tr>
<tr>
<td>Twisted for ideal loading, no proprotor</td>
<td>0.0173</td>
</tr>
<tr>
<td>Untwisted, with proprotor, fixed wake</td>
<td>0.0133</td>
</tr>
<tr>
<td>Twisted for ideal loading, with proprotor, fixed wake</td>
<td>0.0130</td>
</tr>
<tr>
<td>Untwisted, with proprotor, relaxed wake</td>
<td>0.0128</td>
</tr>
<tr>
<td>Twisted for ideal loading, with proprotor, relaxed wake</td>
<td>0.0125</td>
</tr>
</tbody>
</table>

The ideal lift distributions determined for the non-planar wing with and without the proprotor are shown in Fig. 7.5. In the planar section, the normal force is equivalent to the lift force, while on the winglet the normal force is oriented perpendicular to the free-stream and plane of the winglet. The proprotor effects significantly reduce the outboard loading and the normal force on the winglet, which is beneficial from a structural standpoint, as it reduces the root bending moment. It is also noteworthy that the method used to identify the ideal loading forces the lift to zero at the juncture between the wing and winglet. This appears to be due to the discontinuity in the shape of the trailing vorticity because of the dihedral break.
The twist distributions for both ideally loaded cases are shown in Fig. 7.6. The twist at the transition between the wing and winglet is extremely negative, as the ideal loading at that location is forced to zero, as previously discussed. Otherwise, it is notable that similar to the planar wing case, the twist for ideal loading is heavily influenced by proprotor effects.
The vortex-induced drag calculated for the non-planar wing configurations is given in Table 7.3. Without the proprotor, the drag is reduced by approximately 6% as compared to the planar wing. Twisting to achieve an ideal lift distribution reduces the drag by nearly 5% over the untwisted case for the wing without the proprotor. Introducing the proprotor into the system results in a similar magnitude of reduction of vortex-induced drag to that seen in the planar case. Ideal loading, on the other hand, seems to have a larger effect for the non-planar case resulting in a 8% reduction in vortex-induced drag when calculated with a fixed wake and a 5% reduction when calculated with a relaxed wake.
Table 7.3: Comparison of vortex-induced drag coefficients for an untwisted and twisted non-planar LCTR2 wing.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>$C_{D,i}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Untwisted, no proprotor</td>
<td>0.0163</td>
</tr>
<tr>
<td>Twisted for ideal loading, no proprotor</td>
<td>0.0156</td>
</tr>
<tr>
<td>Untwisted, with proprotor, fixed wake</td>
<td>0.0127</td>
</tr>
<tr>
<td>Twisted for ideal loading, with proprotor</td>
<td>0.0117</td>
</tr>
<tr>
<td>Untwisted, with proprotor, relaxed wake</td>
<td>0.0120</td>
</tr>
<tr>
<td>Twisted for ideal loading, with proprotor, relaxed wake</td>
<td>0.0114</td>
</tr>
</tbody>
</table>

A more qualitative assessment of the difference between fixed- and relaxed-wake analysis is visible in the streamlines in a horizontal plane near the root of the winglet at a single time-step, as shown in Fig. 7.7. While the differences between these results are small, they are certainly present. Due to the sensitivity of winglet performance to incoming flow angles (particularly at the root of the winglet) as discussed in Chapter 1, these small differences in the streamlines may necessitate changes in design.
It can be concluded from this study that proprotor effects significantly alter both the ideal loading and the twist required to achieve it for both the planar and non-planar tiltrotor wing. In both cases there was slightly more to be gained in terms of reduced induced drag from optimizing with proprotor effects than without. Wake relaxation was shown to influence the magnitude of the predicted effect in the non-planar case.

### 7.2.3 Twist Optimization with Evolutionary Algorithm

As has been discussed at length in this thesis, the strength of the HOFW method is that the computational costs (both in geometry definition and in finding a solution) are low enough that it can be used to thoroughly explore the design space. As a proof of concept, the planar wing - proprotor concept was investigated with a parallelized evolutionary algorithm with the objective of minimizing drag. This,
like many other aerodynamic optimization problems, is challenging because of the non-linear nature of the design space and the plethora of local minima. As such, it must be emphasized that this study is a proof of concept only and a more sophisticated approach to this problem is certainly possible.

The evolutionary algorithm used was the Covariance Matrix Adaptation Evolutionary Strategy (CMAES) [90]. This algorithm was selected because it is useful for non-linear, non-convex optimization problems. The design variables investigated were wing twist at 10 stations per half-span. The twist was assumed to be symmetric. The bounds on the twist were selected to be +/- 5 degrees. The fitness function was the total drag at the cruise lift. To implement the cruise lift constraint without a penalty function, the angle of attack was adjusted to achieve the desired lift with the input twist distribution. This adjustment typically required 2 or 3 iterations per fitness function evaluation.

The population size per generation was selected to be 10, and 11 total processors were used allowing for one processor per population member per generation. Typically, convergence was reached between 100 and 200 generations, where each generation took approximately 24 minutes. This time was kept low by using fixed-wake analysis and only running to 30 time-steps. The total computational time was therefore on average about 60 hrs or 2.5 days. An example of the convergence of the fitness function as a function of generation number is shown in Fig. 7.8.
Figure 7.8: Convergence of the CMAES algorithm for identification of a minimum drag condition for the planar LCTR2 wing.

The CMAES optimized loading both with and without profile drag are shown in Fig. 7.9. Two aspects of these results are notable. First, the CMAES optimized lift distribution comes remarkably close to reaching the ideal loading condition. This serves as both a verification of the ideal loading method and a promising result from the optimization perspective. The second notable aspect of Fig. 7.9 is that the optimized lift distribution without profile drag is very close to that with profile drag. This seems counter intuitive, but is a result of the fact that the airfoil drag polar for this particular wing is very flat in the $c_l$ range in which the wing is operating, and as a result small changes in local lift coefficient have very little effect on the integrated profile drag.

Although the lift distribution is satisfactory, the twist distribution resulting in
the optimized case is oscillatory and impractical. This is not particularly surprising, given that within the bounds of the model and design parameters, there are an infinite number of twist distributions that would return something very close to the ideal lift distribution. To eliminate this problem, a more sophisticated approach to defining the twist distribution is required.

The drag coefficients (separated by contribution) of the baseline and optimized wing are provided in Table 7.4. Through twisting of the wing, a 3.1% reduction in drag can be achieved. These results should be taken with caution due to the previously discussed unrealistic twist distribution required to achieve them. Regardless, the CMAES optimized induced drag coefficient is within 2 counts of that found through iterative twisting.

Figure 7.9: CMAES optimized loading of the planar LCTR2 wing with and without profile drag.
7.2.4 Summary of Results

The results of these studies in terms of the LCTR2 design show only modest improvements over the baseline. The reduction in wing drag of 3 counts resulting from ideal loading is very small, particularly when considered as a portion of the total vehicle drag. If the baseline cruise L/D of 11.1 listed in Table 7.1 is used to estimate the full vehicle drag, this reduction would only increase the cruise L/D to 11.15. The added structural complexity (and accompanying cost) of implementing a design such as this likely outweighs the very modest benefit. The non-planar wing design was shown to reduce the vortex-induced drag by between 14 and 16 counts, but will also be slightly reduced due to the added wetted area of the winglet. This reduction would result in an increase in cruise L/D to between 11.3 and 11.4. Considering the crude approach taken to the design of the “winglet” in this study, it stands to reason that a more significant reduction could be achieved. For example, a true winglet design would require adjustment of the planform (chord distribution) along with the twist distribution. This adjustment would certainly increase the potential gains over the twist-only optimization performed in this study. In conclusion, the results support the evolution of the LCTR design into its current form, with the most promising path towards further drag reduction in the design of a more sophisticated winglet.

<table>
<thead>
<tr>
<th>Case</th>
<th>$C_{D,i}$</th>
<th>$C_{D,\text{p}}$</th>
<th>$C_D$</th>
<th>% Reduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>0.0133</td>
<td>0.0058</td>
<td>0.0191</td>
<td>–</td>
</tr>
<tr>
<td>Optimized</td>
<td>0.0127</td>
<td>0.0058</td>
<td>0.0184</td>
<td>3.1</td>
</tr>
</tbody>
</table>

Table 7.4: Comparison of drag coefficients for CMAES optimally twisted and untwisted planar LCTR2 wing.
More important than the results themselves are the examples provided by the studies of the types of analysis enabled by the HOFW method. The analytical result known a priori was readily identified by the method, despite presenting only a shallow minimum relative to the baseline design. The influence of wake relaxation was easily identified in all cases. Finally, the simplicity in geometry definition and speed of computation of the HOFW method was highlighted by its use with a genetic algorithm.

7.3 Investigation of Propeller Parameters for Distributed Electric Propulsion

7.3.1 Background on LEAPTech

One innovative technology being developed by NASA uses distributed electric propulsion to achieve a drag reduction [18, 91]. In this approach, distributed electric propulsion is used both to provide thrust and to increase the dynamic pressure on the wing such that the stall-speed requirements can be met with a smaller (and consequently, less drag generating) wing. This technology is referred to as LEAPTech (leading-edge asynchronous propellers technology).

The requirements for the LEAPTech design are provided in Table 7.5. With the baseline design, these requirements require a stall lift coefficient of 4.3 and a cruise lift coefficient of 0.77 (as calculated using free-stream conditions). The propellers each have a diameter of 1.465 ft. The cruise $L/D$ of this first iteration of the LEAPTech aircraft was estimated to be approximately 20 [18].
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Requirement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seating Capacity</td>
<td>4</td>
</tr>
<tr>
<td>Gross Weight</td>
<td>3,000 lb</td>
</tr>
<tr>
<td>Cruise Speed</td>
<td>200 mph</td>
</tr>
<tr>
<td>Cruise Altitude</td>
<td>12,000 ft</td>
</tr>
<tr>
<td>Stall Speed (Sea level)</td>
<td>61 kts</td>
</tr>
</tbody>
</table>

Table 7.5: LeapTECH aircraft [18,91].

The studies that follow are based on a full-scale LEAPTech wing design tested on a truck-based apparatus referred to as the hybrid electric integrated systems testbed (HEIST). Further details of the geometry and operating conditions of this propeller-wing system were graciously made available to the author by the NASA Langley’s Aeronautics Systems Analysis Branch for a separate collaboration, but are not available to be released to the public and are therefore not included in this section. To provide an idea of the general configuration, the lifting surfaces as modeled within the HOFW method are shown in Fig. 7.10.
Figure 7.10: The HEIST configuration as modeled within the HOFW method.

The two studies that follow are provided as examples of potential applications of the method only. Cruise propulsion on the current iteration of the LEAPTech design is to be performed by separate propellers (not included in this model) located at the wingtips, while the inboard propellers (included in the model) would fold against their nacelles in cruise. In contrast, the cruise propulsion in the studies provided in this section is provided by the inboard propellers. This can be considered as an off-nominal operational analysis (as to consider the scenario in which the cruise propellers fail) or a wider exploration of the initial design space. In either case, because of the rapid evolution of the project and design, the results included in this section are not indicative of the current design and should not be considered as such. Instead, they provide interesting (albeit purely academic) examples of the types of analysis possible with the method.

Because of the large number of propellers and large number of blades per propeller, it was necessary to take a few steps to reduce the computational time required for analysis. First, the influence of wake relaxation was neglected (i.e.
the analysis was run with fixed wakes). Second, the resolution on the propeller blades was kept to four panels across the span. In addition, coarse paneling was applied to the wing in comparison with the size of the propellers. Because the paneling was consistent, and the actual geometry of the wing and propeller surfaces was unchanging, it is appropriate to assume that this has little influence on the trends predicted other than to slightly increase uncertainty within the results. These steps resulted in computational times of approximately 4 hours 20 minutes per case.

7.3.2 Investigation of the Influence of Propeller Rotation Direction on Aerodynamic Performance

One interesting design study for the LEAPTech aircraft is an examination of the direction of rotation for the propellers, particularly with regards to how changing this parameter influences the resulting aerodynamic performance of the aircraft. To begin to answer this question, six design alternatives were analyzed within the HOFW method. In these cases, the geometry of the wing is kept constant, and the rotation directions of the propellers (up-inboard vs up-outboard) are varied. A summary of the rotation directions for each case is provided in Fig. 7.11, where a positive one (in yellow) indicates up-inboard rotation and a negative one (in blue) indicates up-outboard rotation.
To run this study, it was necessary to model the propellers at an approximate cruise thrust condition. The thrust required was estimated with the published estimated $L/D$ and weight of the aircraft. The propeller advance ratio was then adjusted to approximately match the estimated thrust.

The resulting wing drag polars are provided in Fig. 7.12. Intuitively, and in agreement with the literature, up-inboard rotation should result in better performance than up-outboard rotation because the induced rotation opposes that resulting from shed vorticity. This is visible in the results as the three lowest drag cases (particularly at with increasing lift coefficient) are Case 1, Case 3, and Case 6, all of which feature more up-inboard rotation than up-outboard rotation.
From a systems perspective, it is more useful to consider the net force in the drag direction including the thrust, in particular in this case because the thrust is influenced by the changes in design. To do so, a non-dimensional coefficient excess thrust, $C_x$, can be defined similar to a drag coefficient, viz.

$$C_x = \frac{T \cos(\alpha) - D}{\frac{1}{2} \rho V^2 \infty S},$$  \hspace{1cm} (7.1)$$

where $S$ is the wing area. Because only the drag of the wing is accounted for in this case, the excess $C_x$ at a given lift condition is physically representative of the allowable drag of the rest of the aircraft such that a trim condition can be met. This coefficient as a function of the lift coefficient augmented with thrust
contributions in the lift direction ($C_L^*$) is provided in Fig. 7.13. The conclusions to be drawn from this figure are largely the same as those found from the previous, in that at the cruise $C_L^*$, the three best performing designs remain Case 1, Case 3, and Case 6.

![Figure 7.13: A comparison of the net non-dimensional force in the drag direction as a function of the net non-dimensional force in the lift direction of each of the six design cases described in Fig. 7.11.](image)

Some tradeoff for the increase in wing efficiency seen in Fig. 7.12 and accompanying increased thrust seen in Fig. 7.13 exists in the form of propeller performance reduction. To elucidate this relationship, the average power coefficient as a function of the augmented lift coefficient is provided in Fig. 7.14. Case 5, for example, which performed poorly in all other metrics, exhibits a relatively low average propeller power coefficient. While this seems attractive from the design perspective,
it is accompanied with a reduced thrust coefficient as well (as can be observed in Fig. 7.13).

Figure 7.14: A comparison of the average propeller power coefficient as a function of the net non-dimensional force in the lift direction of each of the six design cases described in Fig. 7.11.

These tradeoffs can be directly accounted for through a two-dimensional trim of $C_x$ and $C_L$ to determine the minimum power required condition, examples of which are provided in the next section. In this case, conclusions can be drawn directly from the three plots without the need to select a specific $C_x$. The three high performing designs selected from Figs. 7.12 and 7.13 are Cases 1, 3, and 6. Of these three cases, Case 3 requires the lowest power coefficient over the lift coefficients from 0.72 to 1.0. Thus, within the limits of this analysis, uniformly up-inboard propeller rotation appears to provide the highest overall system efficiency
in cruise for this propeller-wing system.

7.3.3 Investigation of the Influence of Propeller Axis Vertical Location

A second design parameter of interest is the vertical location of the propeller axis relative to the wing quarter-chord location. To examine this parameter, seven cases were run with axis locations ranging from one-half radius below the wing to one-half radius above the wing. These cases were run with uniform up-inboard rotation based on the results of the previous study.

The wing drag polars for each vertical axis location are provided in Fig. 7.15. The locations below the wing seem to reduce the wing drag, particularly with increasing lift coefficient.
Figure 7.15: A comparison of the drag polars of each of the seven propeller vertical axis locations considered.

Again, for a more complete picture of the design space, it is necessary to consider the overall system performance, as was done in the previous section. The net force in the thrust direction as a function of the net force in the lift direction are provided in Fig. 7.16, along with the average propeller power as a function of the net force in the lift direction in Fig. 7.17. At the cruise $C_L$, there is an increase in net thrust with decreasing propeller axis location at the same operating conditions. Because this effect is much more pronounced than the reduction in wing drag seen in Fig. 7.15, it is attributable mostly to an increase in propeller thrust due to operation in the wing upwash. The consequence of this increase in thrust is a significant increase in propeller power required, as is clear from Fig. 7.17.
Figure 7.16: A comparison of the net non-dimensional force in the drag direction as a function of the net non-dimensional force in the lift direction for each of the seven propeller vertical axis locations considered. In this comparison, the propellers are kept at constant operating conditions.
A few conclusions can be drawn from these data. First, these trends would not be visible if mutual interaction between the propeller and wing were neglected. In this case, the isolated propeller would have constant thrust and torque, as it is being held at a constant operating condition. Second, in order to parse useful information from this design study, it is necessary to trim both lift and $C_x$ to a single condition and compare the resulting averaged required power coefficients. While this will provide conclusions with respect to this particular propeller design, it also must be noted that there are multiple ways to adjust the thrust (through pitch manipulation and changes in advance ratio), and changes in operating conditions can significantly influence the resulting induced velocity distribution.

Figure 7.17: A comparison of the average propeller power coefficient as a function of the net non-dimensional force in the lift direction for each of the seven propeller vertical axis locations considered.
In this case, a coarse sweep of propeller pitch angles were considered and two-dimensional linear interpolation was used to find the approximate average propeller power coefficient for the cruise lift coefficient and a selected $C_x$ value of 0.02 to approximately match the predicted aircraft cruise $L/D$ of 20. An example of this type of calculation is provided in Fig. 7.18. The resulting trimmed average power coefficient as a function of vertical propeller location is provided in Fig. 7.19.

Figure 7.18: An example of the processed used to determine the trimmed average propeller power coefficient for a set lift and $C_x$ of a given HEIST wing design.
According to Fig. 7.19, placement of the propeller axis below the wing location results in the lowest power requirements in the cruise condition, with the highest power requirements at the two highest vertical locations considered. This trend remains constant for a range of $C_x$ values within the bounds of interpolation between cases considered.

### 7.4 Summary

Two design studies were conducted with the HOFW method to provide examples of its applicability and utility. The first study examined the twist distribution on a large civil tiltrotor design both with and without a winglet. The second study
examined the influence of both rotation direction and vertical location of the 9 propellers on a distributed electric propulsion vehicle. These studies emphasized the unique analytical capabilities of the method along with its computational speed, both in terms of set-up and run-time.
Summary, Future Work, and Conclusions

8.1 Summary

A new higher-order free-wake (HOFW) method has been developed that enables the exploration of the conceptual design space of non-conventional propeller-wing systems. The potential-flow method uses higher order elements to represent the wings and propeller blades as thin lifting surfaces. These higher order elements allow for better force resolution and more intrinsically computationally stable wakes than a comparable vortex-lattice method, while retaining the relative ease of geometric representation inherent to such methods. Both the propeller and wing surfaces are modeled within the same flow field, thus accounting for mutual interactions without the need for empirical models. The flow-field is developed using a time-stepping approach in which the kinematics of the surfaces are dictated individually to allow for simultaneous full system translation and propeller rotation. Enforcing flow tangency at specified control points for each time
step allows for identification of time-accurate variations due to surface-surface and surface-wake interactions. Additional unsteady effects are accounted for within the method in the form of shed circulation and apparent mass effects. Local lift and drag are calculated using the Kutta-Joukowski law with additional unsteady terms in the lift calculation. The influence of the resolution used to model the lifting surfaces on the time-averaged integrated forces was shown to be small above minimum constraints.

To investigate the ability of the HOFW method to predict time-accurate responses, two comparisons were made. First, the method was used to predict the response to a sharp-edged gust and the results compared favorably with the Küßner function. Second, a propeller-wing system was modeled with both the present method and fully unsteady CFD. The time-accurate and frequency responses in lift on the wing were compared. While the method is capable of predicting the lift response with some degree of accuracy, the study confirmed that the unsteady effects experienced by the wing in propeller-wing systems are small. In addition, the resolution needed to resolve these effects within the HOFW method discourages its use in design applications. Still, valuable insights into the interaction itself were gained through the study.

For further validation, time-averaged results found using the HOFW method were compared with experiments and other analysis methods. In the first comparison, the HOFW method was used to predict the performance of a propeller and proprotor. In agreement with the assumptions of the method described in Chapters 2 and 3, decent agreement was found for lightly loaded conditions with errors presenting at high pitch angles and low advance ratios (stall conditions). Two simple propeller-wing systems were next analyzed using the HOFW and two
accepted semi-empirical methods over a range of angles of attack. While the trends were similar, there were clear differences in the results. The two semi-empirical methods were then compared with experimental results and the CFD for a second propeller-wing system, where the propeller was sized such that a large portion of the wing was in the propeller slipstream. In terms of integrated lift, the CFD best matched the experimental results with the HOFW method incurring only slightly more error. The two semi-empirical methods both exhibited significantly more error across the range of angles of attack. Further comparisons emphasized the ability of the HOFW method to quickly and accurately predict details of the lift and drag distributions on the wing that are not available from the simpler semi-empirical methods. Finally, the HOFW method was successfully used to predict experimental trends relating to the spanwise location of a propeller axis and the influence of propeller rotation direction.

Two design studies were conducted to show the practical utility of the method. The first study investigated the twist distribution on a large civil tiltrotor wing. Two approaches were taken, the first used an iterative twist approach to approximate a known “ideal” lift distribution. The second used an evolutionary algorithm to optimize twist. Both approaches resulted in a reduction in predicted wing drag at a constrained lift coefficient. More importantly, the study demonstrated the type of analysis that the method is capable of performing. A second design study focused on the propellers of a distributed electric propulsion vehicle. The influence of discrete sets of potential propeller rotation directions was considered as well as the influence of the vertical location of the propellers. The results showed interactions between the design parameters considered and the average propeller power required for trim.
8.2 Future Work

Several modifications could be made to improve the HOFW method for propeller-wing systems. The leading cause of error currently unaccounted for in terms of wing performance is the influence of the nacelle. The next likely source of error is the omission of the propeller normal force, which becomes particularly influential when the propeller is at a high angle of attack. A more sophisticated stall model would enable better stall and post-stall performance predictions, as well as the introduction of non-linearity into the lift curve. An improvement to geometrical representation of the lifting surfaces and wakes to reduce leakage between elements would also improve the fidelity of the method.

Although unsteady effects were not particularly influential for the cases considered in this thesis, many improvements to the model can be made in this arena. A treatment of the shed circulation to avoid instability in the wake would allow for both relaxed wake analysis in unsteady mode and shed circulation to be included within the propeller wake. A time-accurate assessment of vortex-induced drag is also of potential use, along with a detailed assessment of the influence of leading-edge suction in unsteady profile drag.

8.3 Conclusions

The objective of this thesis was to develop an aerodynamic analysis method to enable design space exploration for non-conventional propeller-wing systems. This objective has been accomplished through meeting the four sub-objectives outlined in Section 1.5.

The first objective was to show that the method is capable of predicting perfor-
mance of both propellers and wings on a design level including mutual interaction, time-accurate responses, and wake relaxation. These capabilities are inherent to the method as it represents all lifting surfaces and wakes in a single time-stepping potential flow solution. This objective was therefore met through the descriptions of the approach to finding a flow solution and calculating forces from that flow solution that are provided in Chapters 2 and 3.

The second objective was to show that the method is fast, both in terms of the definition of a system geometry and in solving of the flow-field. In addition to the inherent speed of a surface singularity method as compared to CFD for most cases, several modifications were made to improve the time of calculation, including special treatment of the propeller velocities (as discussed in Section 2.4.5) and the wake freezing approach (as discussed in Section 4.5). The design sweeps involved in both applications studies discussed in Chapter 7 were automated in geometry definition and completed over very short time-scales (24 minutes per case on a single processor for the LCTR study and less than five hours per case on a single processor for the distributed propulsion study).

The third objective was to show that the method is accurate, at least in trends with design changes. This was accomplished through comparison with other methods and experimental data in both the time accurate and time-averaged analysis. The time-accurate capabilities of the method were addressed in Chapter 5 through comparison with the Küssner function and comparison of the frequency and magnitude of lift oscillations for a propeller-wing system. The time-averaged predictions were considered in Chapter 6. The method matched experimental propeller efficiencies to within 4% for lightly loaded conditions. Increases in lift coefficient due to interaction with a propeller for a series of wings as analyzed with the HOFW
method matched the average of those predicted with two semi-empirical methods with an average of 6.5% error for a lightly loaded propeller case. A comparison of HOFW predictions of lift for a more non-conventional propeller-wing system with experimental results over a range of angles of attack showed an average difference of 0.04 in lift coefficient. For this system, predictions in thrust and torque also matched experimental results within 5% over a small angle of attack range (±5°). The method was less successful at predicting the magnitude of drag in comparison with experimental results, but was capable of qualitatively matching trends in drag, both with changes in lift and for variations in design.

The fourth objective was to show that the method is numerically robust. This was first accomplished through a description of the method itself, as the wake composed of DVEs induces a continuous velocity distribution. To prove this is the case in practice, variation of the performance predictions as a function of the time and space resolution used in the method were provided in Chapter 4. This analysis showed that over a series of resolutions post convergence, the results were shown to vary by less than 0.6% in wing lift and 2.1% in wing drag for a representative case.

In meeting these four objectives, the method has been shown to be better suited for use in conceptual design of non-conventional propeller wing systems than low-order semi-empirical methods and vortex-lattice models we well as more computationally expensive CFD. The low-order semi-empirical methods of Smelt and Davies and McCormick are useful for analysis of conventional propeller-wing systems, but are unable to correctly predict trends for less conventional systems. In addition, they are limited in their prediction ability in that they can predict lift but not drag and no details of the spanwise distributions of these forces. Vortex-lattice
methods are more capable of resolving lift and drag, but require either solid-core models or careful placement of control points to deal with interactions between singularities and control points. Because of the unique properties of the DVE, these issues are not of concern. In addition, the HOFW method requires fewer panels for the same resolution as a vortex-lattice method, thus reducing relative computational time. Finally, although CFD can unarguably resolve more details of the flow field, in terms of the integrated forces and spanwise distribution of forces, it has not proven significantly more accurate than the HOFW method. Furthermore, it requires more man-hours to grid each design case and more processors and time to solve each flow-field. Thus, and in conclusion, the HOFW method provides advantages in terms of speed, accuracy, and/or stability over currently available methods for applications in conceptual design of propeller-wing systems.
Equations for the Velocity Induced by a DVE

The velocity induced by a DVE on a point in the flow field in the local coordinate system (as defined in Fig. 2.1) is

\[ \nabla_i(\xi_0, \eta_0, \zeta_0) = \nabla_{i,\text{filament}}(\xi_0 + \xi_{LE}, \eta_0, \zeta_0, A, B, C, \varphi_{LE}) + \]
\[ \nabla_{i,\text{filament}}(\xi_0 + \xi_{TE}, \eta_0, \zeta_0, -A, -B, -C, \varphi_{TE}) + \]
\[ \nabla_{i,\text{sheet}}(\xi_0 + \xi_{LE}, \eta_0, \zeta_0, B, C, \varphi_{LE}) + \]
\[ \nabla_{i,\text{sheet}}(\xi_0 + \xi_{TE}, \eta_0, \zeta_0, -B, -C, \varphi_{TE}). \]

Expanding this equation using the three-dimensional Biot-savart law and combining terms in the style of Bramesfeld [8] results in
\[ \nabla_i(\xi_0, \eta_0, \zeta_0) = \frac{1}{4\pi} \begin{bmatrix}
  a_{1\xi, LE} - a_{1\xi, TE} & b_{1\xi, LE} - b_{1\xi, TE} & c_{1\xi, LE} - c_{1\xi, TE} \\
  a_{1\eta, LE} - a_{1\eta, TE} & b_{1\eta, LE} - b_{1\eta, TE} & c_{1\eta, LE} - c_{1\eta, TE} \\
  a_{1\zeta, LE} - a_{1\zeta, TE} & b_{1\zeta, LE} - b_{1\zeta, TE} & c_{1\zeta, LE} - c_{1\zeta, TE}
\end{bmatrix} \begin{bmatrix}
  A \\
  B \\
  C
\end{bmatrix}, \]

where the coefficients are defined as

\[
\begin{align*}
  a_{1\xi} &= -G_{11}\zeta_0, & b_{1\xi} &= -G_{12}\zeta_0, & c_{1\xi} &= -G_{13}\zeta_0, \\
  a_{1\eta} &= G_{11}\zeta_0 \tan(\varphi), & b_{1\eta} &= G_{12}\zeta_0 \tan(\varphi), & c_{1\eta} &= G_{13}\zeta_0 \tan(\varphi), \\
  a_{1\zeta} &= G_{11}(\xi_0 - \eta_0) \tan(\varphi), & b_{1\zeta} &= G_{12}(\xi_0 - \eta_0) \tan(\varphi), & c_{1\zeta} &= G_{13}(\xi_0 - \eta_0) \tan(\varphi), \\
  b_{2\eta} &= -\zeta_0 \sum_{i=1,2,6} G_{2i} b_{2j}, \\
  c_{2\eta} &= -\zeta_0 \sum_{i=1,2,4,5,6} G_{2i} c_{2j}, \\
  b_{2\zeta} &= \sum_{i=1-7} G_{2i} b_{2i}, \\
  c_{2\zeta} &= \sum_{i=1-7} G_{2i} c_{2i}.
\end{align*}
\]

In these coefficients, \( \varphi \) is the sweep of the vortex filament or leading edge of the vortex sheet. The additional \( G \) coefficients are defined as
\[ G_{11} = \frac{a_1 \eta + b_1}{(a_1 c_1 - b_1^2) \tau \eta} |_{\eta_R - \eta_L}, \]
\[ G_{12} = \frac{b_1 \eta + c_1}{(a_1 c_1 - b_1^2) \tau \eta} |_{\eta_R - \eta_L}, \]
\[ G_{13} = \frac{(2b_1^2 - a_1 c_1) \eta + b_1 c_1}{(a_1 c_1 - b_1^2) \tau \eta} + a_1^{-\frac{3}{2}} \ln(a_1^2 \tau \eta + a_1 \eta + b_1) |_{\eta_R - \eta_L}, \]
\[ G_{21} = \left[ \frac{\beta_1}{2 \rho} \ln \mu_1(t) + \frac{\beta_2}{\rho} \mu_2(t) \right]_{t_1}^{t_2}, \]
\[ G_{22} = \frac{1}{\xi_0} \left[ -\frac{\beta_2}{2 \rho} \ln \mu_1(t) + \frac{\beta_1}{\rho} \mu_2(t) \right]_{t_1}^{t_2}, \]
\[ G_{23} = \left[ \frac{1}{a_2} r(t) - \frac{b_2}{a_2^2} \ln \mu_3 \right]_{t_1}^{t_2}, \]
\[ G_{24} = \left[ \frac{1}{\sqrt{a_2^3}} \ln \mu_3 \right]_{t_1}^{t_2}, \]
\[ G_{25} = \left[ \frac{1}{2} \ln(k + t^2 + \zeta_0^2) \right]_{t_1}^{t_2}, \]
\[ G_{26} = \left[ \frac{1}{\xi_0} \tan^{-1} \left( \frac{t}{\xi_0} \right) \right]_{t_1}^{t_2}, \]
\[ G_{27} = t_2 - t_1 \]

where
\[ a_1 = 1 + \tan^2(\varphi), \quad a_2 = 1 + \tan^2(\varphi), \]
\[ b_1 = -(\eta_0 + \xi_0 \tan(\varphi)), \quad b_2 = (\xi_0 - \eta_0 \tan(\varphi)) \tan(\varphi), \]
\[ b_{21} = -(\xi_0 - \eta_0 \tan(\varphi)), \quad b_{22} = \zeta_0^2 \tan(\varphi), \]
\[ b_{23} = 0, \quad b_{24} = -\tan(\varphi), \]
\[ b_{25} = -1, \quad b_{26} = 0, \]
\[ b_{27} = 0, \]
\[ c_1 = \xi_0^2 + \eta_0^2 + \zeta_0^2, \quad c_2 = (\xi_0 - \eta_0 \tan \varphi)^2 + \zeta_0^2, \]
\[ c_{21} = -2(\zeta_0^2 \tan(\varphi) + \eta_0(\xi_0 - \eta_0 \tan(\varphi))), \quad c_{22} = -2(\zeta_0^2(\xi_0 - 2\eta_0 \tan(\varphi))), \]
\[ c_{23} = 2 \tan(\varphi), \quad c_{24} = 2(\xi_0 - 2\eta_0 \tan(\varphi)), \]
\[ c_{25} = -2\eta_0, \quad c_{26} = 2\zeta_0^2, \]
\[ c_{27} = 2, \]
\[ r(\eta) = \sqrt{\eta^2 a_1 + 2\eta b_1 + c_1}, \quad r(t) = \sqrt{t^2 a_2 + 2tb_2 + c_2}, \]

and

\[ t = \eta_0 - \eta. \]
Appendix B

Derivation of the Unsteady Lift through Pressure Integration

The unsteady lift force on a two-dimensional flat plate airfoil is derived in this section through pressure integration. Although pressure integration is not used in the calculation of lift in the method itself, this derivation provides a direct link between the results of such an approach and the terms included in the Kutta-Joukowski approach, providing insight into both the physical source of the terms and the differences between the approaches.

To be consistent with the method itself, the derivation uses three coordinate systems: the global coordinate system (x, y, and z positions and u, v, and w velocities), the aerodynamic coordinate system based on the free-stream velocity (lift, side-force, and drag), and the local surface coordinate system (ξ, η, ζ). These three coordinate systems are shown with respect to a section of a lifting surface in Fig. 3.1 reproduced in Fig.B.1 for convenience. The lifting surface in this derivation begins as a continuous distribution of vorticity in the chordwise direction. In addition, to simplify the mathematics, the three coordinate systems are assumed
to have coincident origins at the spanwise center of the leading edge of the lifting surface, and the y-, side-force, and \( \eta \) axes are aligned.

\[
\begin{align*}
\text{Figure B.1: Angle and coordinate system definitions for deriving the unsteady Kutta-Joukowski theorem.}
\end{align*}
\]

The total integrated circulation, \( \Gamma \), at a point \( p \) on the lifting surface (\( \zeta = 0 \), within the DVE coordinate frame) at time \( t \) is circulation oriented along the span that induces velocities according to the right-hand rule in the chordwise direction. The circulation can be related to the vorticity with

\[
\Gamma(\xi_p, \eta_p, 0, t) = \int_0^{\xi_p} \gamma_\xi(\xi, \eta_p, 0, t) d\xi \quad (B.1)
\]

Assuming Fig. B.1 is a slice in the of the lifting surface at a given spanwise location, the velocity potential at a point \( p' \) is

\[
\phi(\xi_{p'}, \eta_{p'}, t) = \int_0^\zeta \frac{-\gamma_\xi(\xi, 0, t)}{2\pi} \theta(\xi_{p'}, \eta_{p'}, \xi) d\xi \quad (B.2)
\]

according to the definition of a potential vortex. In Eq. B.2, \( \theta \) is the angle of the point \( p' \) from the location along the chord in the lifting surface coordinate system (as shown in Fig. 3.1). Note that due to the difference in sign convention in the definition of a positive potential vortex and the positive vorticity defined in Fig.
B.1, the strength of the vorticity is negative in Eq. B.2.

With a converged vorticity distribution, at a given location on the lifting surface and time step, the difference in pressure between the upper and lower surfaces of the lifting surface can be found using Eq. 2.12 to be

\[
P_l - P_u = \rho \left[ \frac{1}{2} (|\nabla \phi_u|^2 - |\nabla \phi_l|^2) + \left( \frac{\partial \phi_u}{\partial t} - \frac{\partial \phi_l}{\partial t} \right) \right]
\]  

(B.3)

This pressure difference results in a force perpendicular to the surface. In the limit as point \( p' \) approaches the lifting surface, the angle, \( \theta \) on the upper surface limits to 0 to the left of \( \xi_{p'} \) and \( \pi \) to the right. Likewise, on the lower surface, \( \theta \) limits to \( 2\pi \) to the left of \( \xi_{p'} \) and \( \pi \) to the right. The upper and lower surface potential functions are thus

\[
\phi_u(\xi, \eta, 0^+, t) = \int_{\xi_{p'}}^{c} -\gamma_\xi(\xi, \eta, 0, t) \frac{d\xi}{2} + \left[ u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*) \right] \cdot \begin{bmatrix} \xi \\ \eta \\ \zeta \end{bmatrix}
\]

and

\[
\phi_l(\xi, \eta, 0^-, t) = \int_{0}^{\xi_{p'}} -\gamma_\xi(\xi, \eta, 0, t) d\xi + \int_{\xi_{p'}}^{c} -\gamma_\xi(\xi, \eta, 0, t) \frac{d\xi}{2} + \left[ u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*) \right] \cdot \begin{bmatrix} \xi \\ \eta \\ \zeta \end{bmatrix}
\]

(B.5)

In Eq. B.4 and B.5, \( u, v, \) and \( w \) are components of velocity (including free-stream
and induced velocities) in the \(x, y,\) and \(z\) directions. They are transformed into the lifting surface coordinate system with a rotation of \(\alpha^*\) about the \(y\)-axis.

Based on the potential functions, the velocity vectors on the upper and lower surfaces in Eq. B.3 are

\[
\nabla \phi_u(\xi, \eta, 0^+, t) = \begin{bmatrix}
\frac{d}{d\xi} \left( \int_{\xi}^c \left( \frac{\gamma_c(\xi, \eta, 0, t)}{2} d\xi \right) + u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*) \right) \hat{\xi} \\
\frac{d}{d\eta} \left( \int_{\xi}^c \left( \frac{\gamma_c(\xi, \eta, 0, t)}{2} d\xi \right) + v(t) \right) \hat{\eta} \\
\frac{d}{d\zeta} \left( \int_{\xi}^c \left( \frac{\gamma_c(\xi, \eta, 0, t)}{2} d\xi \right) + u(t) \sin(\alpha^*) + w(t) \cos(\alpha^*) \right) \hat{\zeta}
\end{bmatrix}
\]

\(\text{(B.6)}\)

and

\[
\nabla \phi_l(\xi, \eta, 0^-, t) = \begin{bmatrix}
\frac{d}{d\xi} \left( \int_0^{\xi} \left( \frac{\gamma_c(\xi, \eta, 0, t)}{2} d\xi \right) + \int_{\xi}^c \left( \frac{\gamma_c(\xi, \eta, 0, t)}{2} d\xi \right) \right) + u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*) \right) \hat{\xi} \\
\frac{d}{d\eta} \left( \int_0^{\xi} \left( \frac{\gamma_c(\xi, \eta, 0, t)}{2} d\xi \right) + \int_{\xi}^c \left( \frac{\gamma_c(\xi, \eta, 0, t)}{2} d\xi \right) \right) + v(t) \right) \hat{\eta} \\
\frac{d}{d\zeta} \left( \int_0^{\xi} \left( \frac{\gamma_c(\xi, \eta, 0, t)}{2} d\xi \right) + \int_{\xi}^c \left( \frac{\gamma_c(\xi, \eta, 0, t)}{2} d\xi \right) \right) + u(t) \sin(\alpha^*) + w(t) \cos(\alpha^*) \right) \hat{\zeta}
\end{bmatrix}
\]

\(\text{(B.7)}\)

Both terms in the \(\hat{\eta}\) direction may be assumed to be much smaller than those in the \(\hat{\xi}\) direction. This is akin to assuming that the circulation variation over the chord has a more pronounced effect on the local velocity (and consequently, the local pressure) than the variation of the circulation over the span. Because the gradient of the potential function is physically the velocity, and assuming flow tangency on the lifting surface, the \(\hat{\zeta}\) terms on both the upper and lower surface must also be zero.
This leaves only the $\hat{\xi}$ terms in the final solution. The partial derivative of the first terms of the upper and lower surface $\hat{\xi}$ components are given by

$$\frac{d}{d\xi} \left( \int_{\xi'}^{c} -\frac{\gamma \xi}{2} d\xi \right) = \frac{\gamma \xi}{2}$$ \hspace{1cm} (B.8)

and

$$\frac{d}{d\xi} \left( \int_{0}^{\xi'} -\gamma \xi d\xi + \int_{\xi'}^{c} -\frac{\gamma \xi}{2} d\xi \right) = -\frac{\gamma \xi}{2}$$ \hspace{1cm} (B.9)

respectively, due to the fundamental theorem of calculus. Implementation of these results into the first term on the right hand side of Eq. B.3 yields

$$\frac{1}{2} \left( |\nabla \phi_u|^2 - |\nabla \phi_l|^2 \right) = (u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*)) \cdot \gamma \xi(\xi,\eta,t)$$ \hspace{1cm} (B.10)

Returning to the original Eq. B.3, the last term on the right hand side can be addressed as follows,

$$\frac{\partial \phi_u}{\partial t} - \frac{\partial \phi_l}{\partial t} = \frac{\partial}{\partial t} (\phi_u - \phi_l)$$

$$= \frac{\partial}{\partial t} \left[ \int_{\xi'}^{c} \frac{-\gamma \xi}{2} d\xi - \left( \int_{0}^{\xi'} -\gamma \xi d\xi + \int_{\xi'}^{c} \frac{-\gamma \xi}{2} d\xi \right) \right]$$ \hspace{1cm} (B.11)

Substitution of Eqs. B.10 and B.11 into Eq. B.3 results in

$$P_l - P_u = \rho \left[ (u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*)) \cdot \gamma \xi + \frac{\partial \Gamma(\xi,\eta,t)}{\partial t} \right]$$ \hspace{1cm} (B.12)
Integration of Eq. B.12 over the chord gives the total force per unit span in the \( \zeta \) direction of the lifting surface reference frame, viz.

\[
\overline{F}_\zeta' = \rho \int_0^c \left[ \left( u(t) \cos(\alpha^*) - w(t) \sin(\alpha^*) \right) \cdot \gamma_\xi + \frac{\partial \Gamma(\xi, t)}{\partial t} \right] d\xi \tag{B.13}
\]

The first term on the right hand side of Eq. B.13 accounts for quasi-steady lift and the influence of circulation shed into the wake via the implementation of time dependent boundary conditions. The second term on the right hand side of Eq. 3.2 accounts for non-circulatory effects.
Appendix C

Derivation of Kutta-Joukowski Theorem for Vortex-Induced Drag

Calculation of Single, Multiple, and Intersecting Wakes

C.1 Single Wake

The control volume shown in Fig. C.1 contains a wing modeled in the potential flow domain with distributed vorticity in both the spanwise and chordwise direction. The vorticity on the wing is shed into the wake, which is modeled as rigid and drag free. The control volume is large enough that on all outer surfaces other than the Trefftz plane, the influence of the wing and wake are negligible.
Application of a momentum balance to the this control volume, results in the classical Trefftz plane formula for vortex-induced drag [44], viz.

\[ D_i = \frac{\rho}{2} \int \int_{TP} (v^2 + w^2 - u^2) dydz \]  

where \( \rho \) is the fluid density, \( TP \) indicates the Trefftz Plane (isolated in Fig. C.2), and \( u, v \) and \( w \) are the induced velocities in the streamwise (\( x \)), spanwise (\( y \)) and vertical (\( z \)) directions respectively.
Assuming that the vorticity in the wake is aligned with the x-axis, it can be assumed that $u^2 << v^2, w^2 [44]$. In this case, Eq. C.1 can be recast in terms of the potential function, $\phi$ as

$$D_i = \rho \frac{1}{2} \int_{TP} \left( \frac{\partial \phi^2}{\partial y} + \frac{\partial \phi^2}{\partial z} \right) dydz,$$

which is equivalent to

$$D_i = \rho \frac{1}{2} \int_{TP} \left( \nabla \phi \cdot \nabla \phi \right) dydz. \quad (C.3)$$

Assuming that the potential function satisfies the Laplace equation, and via the implementation of a vector identity, Eq. C.3 can be rewritten as

$$D_i = \rho \frac{1}{2} \int_{TP} \left( \nabla \cdot \phi \nabla \phi \right) dydz. \quad (C.4)$$

Eq. C.4 can be converted from a surface integral to a contour integral via the
Divergence Theorem, resulting in

\[ D_i = \frac{\rho}{2} \oint_C \left( (\phi \nabla \phi) \cdot \hat{n} \right) dl \] (C.5)

where the contour \( C \) is that surrounding the domain, as defined in Fig. C.2, and \( \hat{n} \) is the vector normal to the contour (to the outside).

Because the outside contour is assumed to be large enough that there are no perturbations therein, it can be neglected. The inner contour section (the section surrounding the wake surface) can be assumed to be nearly infinitely thin, and can thus be calculated via two line integrals - one line directly below the wake vortex sheet (L) and one line directly above the wake vortex sheet (U). With these assumptions, Eq. C.5 can be reduced to

\[ D_i = \frac{\rho}{2} \left[ \int_{-b/2}^{b/2} (\phi_L \nabla \cdot \hat{k}) dl_L + \int_{-b/2}^{b/2} (\phi_U \nabla \cdot \hat{k}) dl_U \right] \] (C.6)

If it is assumed that the velocity normal to the vortex sheet is continuous and because the scalar line integral is independent of orientation, Eq. C.6 can be rewritten under a single line integral as

\[ D_i = \frac{\rho}{2} \int_{-b/2}^{b/2} (\phi_U - \phi_L) \nabla_n dl \] (C.7)

To simplify the explanation, the wake can be considered to be a continuous planar vortex sheet aligned with the plane defined by \( z = 0 \), whose strength varies with location along the \( y \)-axis (\( \gamma(y) \)). The potential function at a point \( P \) as defined by Fig. C.3 due to the vortex sheet is

\[ \phi(y_P, z_P) = -\int_{-b/2}^{b/2} \frac{\gamma(y)}{2\pi} \theta(y_P, z_P, y) dy. \] (C.8)
To determine the value of the potential function directly above the vortex sheet at a location \( y' \), Eq. C.8 can be expanded as

\[
\phi_U(y') = -\left[ \int_{-b/2}^{y'} \frac{\gamma(y)}{2\pi} \, dy + \int_{y'}^{b/2} \frac{\gamma(y)}{2\pi} \, dy \right]. \tag{C.9}
\]

Likewise, the value of the potential function directly below the vortex sheet at \( y' \) is

\[
\phi_L(y') = -\left[ \int_{-b/2}^{y'} \frac{\gamma(y)}{2\pi} \, dy + \int_{y'}^{b/2} \frac{\gamma(y)}{2\pi} \, dy \right]. \tag{C.10}
\]

Using these two equations, the value of \( \phi_U - \phi_L \) from Eq. C.7 can be rewritten as

\[
\phi_U(y') - \phi_L(y') = \int_{b/2}^{y'} \gamma(y) \, dy, \tag{C.11}
\]

which is by definition equivalent to the value of the bound vorticity \( \Gamma \) at location \( y' \). Combining Eq. C.7 and Eq. C.11 gives the result for induced drag

\[
D_i = -\frac{\rho}{2} \int_{-b/2}^{b/2} \Gamma(y) \nabla_n(y) \, dy. \tag{C.12}
\]

The Kutta-Joukowski theorem as applied at the trailing edge of a simple planar wing for the calculation of induced drag is
where $V_n$ is the down-wash induced by the trailing vortex sheet. This is identical to Eq. C.12 aside from a factor of two resulting from the difference between the down-wash induced by a semi-infinite vortex sheet at the lifting line versus an infinite vortex sheet in the Trefftz plane. Calculation of induced drag at the trailing edge with this formulation of the Kutta-Joukowski theorem and in the Trefftz plane are thus equivalent for the single planar wing with a drag-free wake.

### C.2 Multiple and Intersecting Wakes

To further address the issue of induced drag for the DVE system, it must be addressed that the upstream propeller is not actually an actuator disk, but instead a lifting surface with a finite wake. To do so, the induced drag of a system of lifting surfaces with non-intersecting wakes must be derived first, following a similar process as with a single lifting surface. Then, the induced drag of a system of lifting surfaces with intersecting wakes will be derived.

The example geometry used in this derivation, as shown in Fig. C.4, is mathematically simplified in that it consists of two wake surfaces of equal span, each parallel to the x-y plane. The result is more generally valid for any system of lifting surfaces with non-intersecting wakes. In the following analysis, the upper wing will be referred to as wing one, while the lower wing will be referred to as wing two to differentiate the wing references from the upper and lower surface references.
The derivation for induced drag of a single wing through Eq. C.5 remains valid for the two wing system. The contour shown in Fig. C.4 that encloses the two wakes is piecewise continuous. A line integral over a piecewise smooth curve is the sum of the integrals over the smooth sections [92]. Thus, in a similar manner to Eq. C.6, the contour integral from Eq. C.5 can be split into four line integrals, resulting in the general form

$$D_i = \frac{\rho}{2} \left[ \int_{-b/2}^{b/2} (\ldots) \, dl_{U1} + \int_{-b/2}^{b/2} (\ldots) \, dl_{L1} + \int_{-b/2}^{b/2} (\ldots) \, dl_{U2} + \int_{-b/2}^{b/2} (\ldots) \, dl_{L2} \right], \quad (C.14)$$

where the ellipsis represents \((\phi \nabla \phi) \cdot \hat{n}\) as evaluated along each line. As \(\nabla \phi\) is, by definition of the potential function, the velocity at the point of interest, the integrals over the first wing can be rewritten as
\[
\int_{-b/2}^{b/2} \left( (\phi_{L1} \nabla) \cdot \hat{k} \right) dl_{L1} + \int_{-b/2}^{b/2} \left( (\phi_{U1} \nabla) \cdot -\hat{k} \right) dl_{U1}.
\]  
(C.15)

As in the single wing case, Eq. C.15 can be simplified to

\[
\int_{-b/2}^{b/2} - (\phi_{U1} - \phi_{L1}) \nabla_n dl_{L1}.
\]  
(C.16)

To simplify further, it is necessary to expand \( \phi_{U1} \) and \( \phi_{L1} \). Assuming that the two wakes are located in planes defined by \( z = z_1 \) and \( z = z_2 \) respectively, the value of the upper and lower potential function at location \( y' \) on the first wing can be calculated as

\[
\phi_{U1}(y') = -\left[ \int_{-b/2}^{y'} \frac{\gamma_1(y)}{2\pi} (0) dy + \int_{y'}^{b/2} \frac{\gamma_1(y)}{2\pi} dy \right] + \int_{-b/2}^{b/2} \frac{\gamma_2(y)}{2\pi} \theta(y', z_1, y) dy,
\]  
(C.17)

and

\[
\phi_{L1}(y') = -\left[ \int_{-b/2}^{y'} \frac{\gamma_1(y)}{2\pi} (2\pi) dy + \int_{y'}^{b/2} \frac{\gamma_1(y)}{2\pi} dy \right] + \int_{-b/2}^{b/2} \frac{\gamma_2(y)}{2\pi} \theta(y', z_1, y) dy.
\]  
(C.18)

Because the last term in Eq. C.17 is equivalent to the last term in Eq. C.18, it cancels when the two terms are subtracted as in Eq. C.16. The resulting integral is thus nearly identical to the single wing case, i.e.

\[
\int_{-b/2}^{b/2} -\Gamma_1(y) \nabla_n (y, z_1) dy.
\]  
(C.19)

Likewise, the integral for the lower wing can be shown to result in
\[ \int_{-b/2}^{b/2} -\Gamma_2(y)\nabla_n(y, z_2) dy. \] (C.20)

Using these results, the integrated drag for the full system is

\[ D_i = -\frac{D}{2} \left[ \int_{-b/2}^{b/2} \Gamma_1(y) \nabla_n(y, z_1) dy + \int_{-b/2}^{b/2} \Gamma_2(y) \nabla_n(y, z_2) dy \right]. \] (C.21)

This result indicates that the induced drag of the combination of wings with non-intersecting wakes can be calculated as the sum over all of the wings of the product of the bound vorticity on a wing at a given spanwise location and the total velocity perpendicular to that surface at that spanwise location in the Trefftz plane. It is important to note that as this analysis is taking place in the Trefftz plane, the velocity induced by the bound vorticity on either lifting surface is not included in the normal velocity.

A system of two lifting surfaces with intersecting wakes is shown in Fig. C.5.
Figure C.5: The intersecting wakes of two planar lifting surfaces as shown in the Trefftz plane.

The contour enclosing the wakes is piecewise smooth, as was the case for the non-intersecting wakes. Thus, in a manner similar to that used for non-intersecting wakes, the contour shown in Fig. C.5 can be split into eight line integrals, viz.

\[
D_i = \frac{\rho}{2} \left[ \int_{-b/2}^{l_{int}} (...) \, dl_{U1,L} + \int_{-b/2}^{l_{int}} (...) \, dl_{L1,L} \\
+ \int_{l_{int}}^{b/2} (...) \, dl_{U2,L} + \int_{l_{int}}^{b/2} (...) \, dl_{L2,L} \\
+ \int_{l_{int}}^{b/2} (...) \, dl_{U1,R} + \int_{l_{int}}^{b/2} (...) \, dl_{L1,R} \\
+ \int_{l_{int}}^{b/2} (...) \, dl_{U2,R} + \int_{l_{int}}^{b/2} (...) \, dl_{L2,R} \right]
\]  
(C.22)

where \( l_{int} \) is the location of the intersection between the two wakes along the line integral and the ellipsis (as before) represents \((\phi \nabla \phi) \cdot \hat{n}\) as evaluated along each line. Because the point of intersection lies on the boundary between smooth
curves, the results from the non-intersecting wake case can be applied directly to the intersecting wake case. In summary, the total drag of the system is the sum over all of the surfaces of the local bound vorticity and the induced velocity normal to the surface in the Trefftz plane.

Calculation of induced drag with the Kutta-Joukowski method and in the Trefftz plane are equivalent for the two lifting-surface cases if the trailing edges are located at the same streamwise position as in an unstaggered biplane configuration. In this case, application of Eq. C.13 to the full system results in

\[ D_i = \rho \int_{-b/2}^{b/2} \Gamma_1(y) V_n(y) dy + \rho \int_{-b/2}^{b/2} \Gamma_2(y) V_n(y) dy, \]  

(C.23)

where \( V_n \) at the lifting line includes both self-induced and mutually induced velocities due to the semi-infinite vortex sheets. Again, the difference between Eq. C.23 and Eq. C.21 is a result of the induced velocities in C.23 being half of those in Eq. C.21 due to the semi-infinite versus infinite vortex sheet induction.

The case of staggered lifting surfaces results in the same aggregate induced drag as a direct result of Munk’s stagger theorem. This is clear in the Trefftz plane result as there is no influence of the streamwise position of the lifting surfaces. For the trailing edge calculation, the influence of the bound vorticity in terms of mutual interaction can no longer be neglected. Thus, although the total vortex-induced drag of this system is equivalent to that given in Eq. C.23, the drag on each individual wing reflects the influence of the bound vorticity and offset in semi-infinite vortex sheets. These influences are thus equal and offsetting for a fixed circulation distribution and a fixed, planar, drag-free wake.

In summary, calculation of induced drag at the trailing edge using the Kutta-Joukowski theorem is consistent with Trefftz plane results if the velocity normal
to the surface used is that due to all wakes. When calculating at the trailing edge, this also includes the influence of the bound vorticity of any other lifting surface to delineate the induced drag of each individual surface. It is worth noting that the results of the two lifting-surface cases are in agreement with results of Prandtl and Tietjens [93] for biplanes.
Appendix D

Trefftz Plane Analysis of Propeller-Wing System

A control volume of inviscid, irrotational, incompressible fluid is defined as shown in Fig. D.1, where the solid surfaces (propeller disk and wing) are subdivisions of the control volume. This subdivision of the control volume follows the approach taken by Veldhuis [3] and allows the thrust and drag to be included as pressures forces.
The conservation of momentum [63] applied to the control volume gives
\[
\iint \int_V \frac{\delta}{\delta t} (\rho \vec{U}) dV + \iint_S \rho \vec{U} (\vec{U} \cdot \hat{n}) dS = - \iint_S p\hat{n} dS + \iint \int_V \rho \vec{F} dV \quad (D.1)
\]

The first term (from left to right) is the rate of change of momentum within the control volume. Due to the periodic-unsteady nature of the flowfield (as discussed in Section 3.3.1.1), the time-averaged flow can be assumed steady for this analysis. The second term in Eq. D.1 is the momentum flux over the control volume surface. The third and fourth terms are the net pressure and body forces respectively. In this analysis, the thrust and torque exerted by the actuator disk are treated as pressure terms, although it must be noted that this implies that the actuator disk is better described as a time-averaged propeller, with the ability to exert velocities not only in the \( u \) direction, but \( v \) and \( w \) as well. Finally, there are no body forces in this analysis. Implementing these assumptions, Eq. D.1 can be written as

\[
\iint_S \rho \vec{U} (\vec{U} \cdot \hat{n}) dS = - \iint_S p\hat{n} dS \quad (D.2)
\]

where \( S \) includes the wing and propeller surfaces as well as the outer surfaces of the control volume.

In the \( x \)-direction, the pressure differential over the surface of the propeller is equivalent to the thrust while the surface pressure differential on the wing is the induced drag of the wing. The additional term \( D_{int} \) refers to the interference drag due to the interaction between the propeller and wing.

Assuming that the side cylinder and entrance planes are far away from the propeller and wing, Eq. D.2 can be rewritten as
\[
D_i + D_{\text{int}} - T + \iint_{TP} (p - p_{\infty})dS = - \iint_{TP} \rho u(U_{\infty} + u)dS \quad \text{(D.3)}
\]

With the exception of thrust and interference drag, this result is the same as that obtained in the traditional Trefftz plane analysis. The introduction of thrust and torque into the control volume by the actuator disk increases the total energy of the flow within the streamtube. In other words, the total (stagnation) pressure within the streamtube is not equal to the total pressure outside of the streamtube. This addition calls into question whether or not traditional Trefftz plane induced drag results are valid. To investigate this system, the Trefftz plane can be split into an outer wake area \(A_O\) and a streamtube wake area \(A_{ST}\) as shown in Fig. D.1. With this separation, Eq. D.3 can be rewritten

\[
D_i + D_{\text{int}} - T = - \iint_{A_{ST}} (p_* - p_{\infty*})dA_{ST} - \iint_{A_{ST}} \rho (u_* + u_p)(U_{\infty} + u_p + u_*)dA_{ST} \\
+ - \iint_{A_O} (p - p_{\infty})dA_{O} - \iint_{A_O} \rho u(U_{\infty} + u)dA_{O} \quad \text{(D.4)}
\]

where \(p_{\infty*}\) is the pressure in the streamtube in the far wake. The area of the streamtube adjusts such that in the far wake, there is no static pressure differential between the outer wake and the streamtube. Thus, \(p_{\infty*}\) is equivalent to \(p_\infty\) and can be calculated as

\[
p_\infty = P_{0,st} - \frac{1}{2} \rho \left[ (U_{\infty} + u_p)^2 + v_p^2 + w_p^2 \right]. \quad \text{(D.5)}
\]

Finally, \(p_*\) is the perturbation in pressure due to additional velocity variation in the streamtube (due to the wing for example), and can be calculated according to
Bernoulli’s principle for incompressible flow inside the streamtube as

\[ p_* = P_{0, st} - \frac{1}{2} \rho \left[ (U_\infty + u_p + u_*)^2 + (v_p + v_*)^2 + (w_p + w_*)^2 \right] \]  \hspace{1cm} (D.6)

Working through the right hand terms in Eq. D.4, \((p_* - p_\infty)\) can be expanded according to the Bernoulli equation, viz.

\[ p_* - p_\infty = \left( P_{0, st} - \frac{1}{2} \rho \left[ (U_\infty + u_p + u_*)^2 + (v_p + v_*)^2 + (w_p + w_*)^2 \right] \right) - \left( P_{0, st} - \frac{1}{2} \rho \left[ (U_\infty + u_p)^2 + v_p^2 + w_p^2 \right] \right) \]  \hspace{1cm} (D.7)

Further expansion of the polynomials and canceling of terms in Eq. D.7 results in

\[ p_* - p_\infty = -\frac{1}{2} \rho \left[ 2u_*U_\infty + 2u_p u_* + u_*^2 + 2v_p v_* + v_*^2 + 2w_p w_* + w_*^2 \right] \]  \hspace{1cm} (D.8)

The second term on the right hand side of Eq. D.4 (the momentum flux within the streamtube) can be expanded as follows

\[ \rho (u_p + u_*) (U_\infty + u_p + u_*) = \rho (U_\infty u_p + U_\infty u_* + u_p^2 + 2u_p u_* + u_*^2) \]  \hspace{1cm} (D.9)

The first two terms on the right hand side of Eq. D.4, when expanded via Eq. D.8 and D.9, results in
\[-\iint_{A_{ST}} (p_* - p_{\infty}) dA_{ST} - \iint_{A_{ST}} \rho (u_* + u_p) (U_{\infty} + u_p + u_*) dA_{ST} =
- \iint_{A_{ST}} \rho \left[ U_{\infty} u_p + u_p^2 + \frac{u_*^2}{2} - \frac{v_*^2}{2} - \frac{w_*^2}{2} + u_p u_* - v_p v_* - w_p w_* \right] \]  

\[(D.10)\]

The terms on the right hand side of Eq. D.10 are physical in nature. The first two terms are equivalent to the thrust according to classical propeller momentum theory. According to this theory translated into the current notation (recall \(A_{ST}\) is not equal to \(A_{disk}\)), the thrust of the propeller is given by

\[T = \rho A_{ST} u_p \left( U_{\infty} + u_p \right) \]  

\[(D.11)\]

Thus, if the stream-wise induced velocities are assumed constant over the disk, the terms are consistent. The three middle terms (* terms) are consistent with the Trefftz plane induced drag on a lifting surface due to the wake of that lifting surface. Thus, the interference drag consists of the final three terms on the right hand side.

Outside of the propeller wake, the theory is unaffected by the presence of the streamtube. Thus the total integrated induced drag on the wing is given by

\[D_i = \iint_{A_{ST}} \frac{1}{2} \rho (v_*^2 + w_*^2 - u_*^2) dA_{ST} + \iint_{A_O} \frac{1}{2} \rho (v^2 + w^2 - u^2) dA_O \]  

\[(D.12)\]

and the total interference drag due to interaction with the propeller is given by

\[D_{int} = \iint_{A_{ST}} \rho (-u_p u_* + v_p v_* + w_p w_*) dA_{ST} \]  

\[(D.13)\]
It should be noted that the losses on the propeller due to swirl are accounted for in the change in total pressure, but do not influence the static pressure balance because it is assumed that the streamtube area adjusts to match the static pressure in the freestream. In addition, although this integral is taken over the streamtube only, the propeller induced velocities would be zero outside of the streamtube, and thus the integrated answer over the full Trefftz plane would be identical.

The two induced drag terms are consistent with the Kutta-Joukowski approach applied at the trailing edge of the wing based on the velocity induced by the wake of the lifting surface only. The interference term must be accounted for separately.

Assuming that the wake induces velocities in the $y$ and $z$ directions, the $u_*$ term is zero, and the first term in Eq. D.13 can be neglected. The remaining terms can be written in terms of their respective potential functions as

$$D_{int} = \int \int_{A_{ST}} \rho (\nabla \phi_w \cdot \nabla \phi_p) dA_{ST} \quad (D.14)$$

where $\phi_w$ is the potential function due to the wake of the wing and $\phi_p$ is the potential function due to the streamtube. Applying a vector identity, Eq. D.14 can be rewritten as

$$D_{int} = \int \int_{A_{ST}} \rho (\nabla \cdot \phi_w \nabla \phi_p) dA_{ST}. \quad (D.15)$$

Eq. D.15 can be then converted to a contour integral over the surface of the wing from a surface integral using the Divergence Theorem, resulting in

$$D_{int} = \rho \oint_c ((\phi_w \nabla \phi_p) \cdot \hat{n}) dl. \quad (D.16)$$

Using the same approach as was used in Appendix C, Eq. D.16 can be converted
to a single line integral, viz.

\[ D_{int} = \rho \int_{-b/2}^{b/2} - (\phi_{w,U} - \phi_{w,L}) \nabla_{p,n} dy \]  \hspace{1cm} (D.17)

Substitution of the bound vorticity of the wing into Eq. D.17 results in the final equation for interference drag,

\[ D_{int} = \rho \int_{-b/2}^{b/2} \Gamma_{w}(y) \nabla_{p,n} dy \]  \hspace{1cm} (D.18)

which is consistent with the results of Veldhuis [3].

Because the tangential induced velocities at the plane of the propeller are the same as in the far wake, there is no conversion necessary from an infinite to semi-infinite wake. Combining all terms, the final equation for drag as calculated at the trailing edge (consistent with both the Kutta-Joukowski at the trailing edge and the Trefftz plane analysis) can be written as

\[ D = \rho \int_{-b/2}^{b/2} \Gamma_{w}(y) (\nabla_{p,n,te} + \nabla_{w,n,te}) dy \]  \hspace{1cm} (D.19)
Appendix E

Approach to Identification of Ideal Loading Condition

Munk’s third theorem [37] states that in the Trefftz plane (down-stream infinity), the induced drag is minimized if the velocity normal to each lifting element is proportional to the cosine of the inclination angle of the element. This theorem is illustrated by Fig. E.1, where $V_n$ is the normal velocity to the element and $\theta$ is the inclination of the element.
A non-planar wing can be modeled with a series of horseshoe vortices, each of strength $\Gamma_j$. In the Trefftz plane there is no contribution from the bound section of the vortex and thus the model can be reduced to trailing vortex pairs of equal strength and opposite rotation [94]. Applying the Kutta-Joukowski law to convert wing loading into circulation, and the Biot-Savart law to calculate induced velocities due to the trailing vortices, the normal induced velocity at each control point can be calculated as

$$V_{n,i} = \sum_{j=1}^{m} \frac{(c_n c_j)}{c_{AV}} A_{ij}$$

(E.1)

where $c_n$ is the normal section load coefficient, $c$ is the local chord, $c_{AV}$ is the average chord and $A_{ij}$ is the geometric influence coefficient. This coefficient is calculated as

Figure E.1: Graphical depiction of Munk’s third theorem.
where $\theta$ is the local inclination angle, $s$ is the semi-width of the vortex pair, and the remaining variables are defined as

\[
y' = (y_i - y_j)\cos(\theta_j) + (z_i - z_j)\sin(\theta_j)
\]
\[
z' = -(y_i - y_j)\sin(\theta_j) + (z_i - z_j)\cos(\theta_j)
\]
\[
R_1 = (z')^2 + (y' - s')^2
\]
\[
R_2 = (z')^2 + (y' + s')^2
\]

According to Munks third theorem, the normal velocity for minimum induced drag is obtained when

\[
\frac{v_{n,i}}{V_\infty} = \frac{w_0}{V_\infty} \cos(\theta_i)
\]

Combining equation E.1 with equation E.3 and converting to matrix notation yields

\[
[\cos(\theta_i)] = \frac{1}{w_0/V_\infty} [A_{ij}] \left( \begin{array}{c} (c_n c)_j \\ c_{AV} \end{array} \right)
\]

which, when solved for the optimum loading becomes
The total wing lift coefficient can then be calculated as

$$C_L = 2 \sum_{j=1}^{m} \left( \frac{(c_n c_j)}{c_{AV}} \right) (s_j) \cos(\theta_j)$$ (E.6)

where \( s \) is the non-dimensional semi-width of the vortex pair. Combining these two equations eliminates the local loading such that the downwash can be determined using only the lift coefficient and wing geometry. Once the downwash is determined, calculating the loading is a simple matrix inversion.

In order to introduce a propeller into this formulation, the additional velocities induced by the propeller in both the axial and tangential direction must be included. To implement these induced velocities, two major assumptions are made. First, it is assumed that the condition for least induced drag with the propeller is, as without the propeller, given by Munk’s third theorem. As mentioned previously, this assumption was made without a rigorous proof. The change in axial velocity can be implemented as a simple scaling of \( \frac{w_0}{V_\infty} \) by the ratio of the freestream velocity to the local velocity \( V_\infty/(V_\infty + V_{ind,axial}) \), to ensure that the downwash (velocity induced normal to the surface) obeys the theorem across the velocity jump. Second, it is assumed that the downwash ratio can be calculated iteratively by assuming that the normal velocity is a sum of the velocity induced by the trailing vortices and the surface normal component of the tangential velocity induced by the propeller. Thus equation E.1 can be rewritten as

$$\frac{V_{n,i}}{V_\infty} = \sum_{j=1}^{m} \left( \frac{(c_n c_j)}{c_{AV}} \right) A_{ij} + \frac{V_{ind,n,i}}{V_\infty}$$ (E.7)
where $V_{\text{ind},n,i}$ is related to the induced tangential velocity and geometry of the wing as depicted in Figure E.2.

The normal velocity is given by the component of the tangential velocity in the surface normal direction and can be calculated as

$$V_{\text{ind},n,i} = \frac{V_{\text{ind},t,i}}{V_\infty} \left[ \sin(\theta) \sin(\psi) + \cos(\theta) \cos(\psi) \right] \quad (E.8)$$

Eq. E.5 is then

$$\left[ \frac{(c_n c)_j}{c_{AV}} \right] = \frac{1}{w_0/V_\infty} [A_{ij}]^{-1} \left[ \cos(\theta_i) - \frac{V_\infty}{w_0} \frac{V_{\text{ind},n,i}}{V_\infty} \right] \quad (E.9)$$

The downwash must then be solved for assuming a given lift coefficient by combining equations E.6 and E.9. Because this combined equation depends on the downwash ratio, the equation must be solved iteratively.
References


Vita
Julia Ann Cole

Julia Ann Cole received her bachelor’s degree in Aerospace Engineering from the Pennsylvania State University in 2007. She earned her Master’s degree in Aerospace Engineering from the Georgia Institute of Technology in 2008. During her doctoral work, she spent time as a teaching assistant, instructor, and research assistant in the Aerospace department. She also spent two years teaching Mechanical Engineering at Bucknell University and one year working for the Applied Research Laboratory.