DEVELOPMENT OF AN IMPROVED LOW-ORDER MODEL FOR

PROPELLER-WING INTERACTIONS

by

Joshua Taylor Goates

A thesis submitted in partial fulfillment
of the requirements for the degree

of

MASTER OF SCIENCE

in

Mechanical Engineering

Approved:

Douglas Hunsaker, Ph.D.  Robert E. Spall, Ph.D.
Major Professor  Committee Member

Barton Smith, Ph.D.  Laurens H. Smith, Ph.D.
Committee Member  Interim Vice President for Research and
                  Dean of the School of Graduate Studies

UTAH STATE UNIVERSITY
Logan, Utah

2018
ABSTRACT

Development of an Improved Low-Order Model for Propeller-Wing Interactions

by

Joshua Taylor Goates, Master of Science

Utah State University, 2018

Major Professor: Douglas Hunsaker, Ph.D.
Department: Mechanical and Aerospace Engineering

An improved low-order method for computationally modeling the effects of prop wash on lifting surfaces is presented. This method combines a traditional propeller blade element model, a novel turbulent prop wash model, and a modern numerical lifting line model to provide accurate results at an efficient computational cost. The traditional propeller blade element model is expanded to increase its robustness and accuracy. The turbulent prop wash model employs observations from the development of turbulent jets of air to model the effects of turbulent mixing on the prop wash development. It is shown that this turbulent prop wash model provides more accurate results at low advance ratios than existing inviscid methods. The prop wash velocities are then added to the local velocities of the numerical lifting line model to show the effects of the prop wash on lifting surfaces. A number of reduction factors are applied to the prop wash velocities to produce accurate results. The results of this model are compared to experimental results. It is shown that this model provides accurate results for the effects of prop wash on lifting surfaces using efficient, low-order methods.
PUBLIC ABSTRACT

Development of an Improved Low-Order Model for Propeller-Wing Interactions

Joshua T. Goates

For aircraft that have propellers mounted in front of the wings or tail, the prop wash produced by the propellers can have a strong influence on the aerodynamics of the aircraft. As the accelerated air from the propeller flows over the wings and tail, it can cause an alteration in the aerodynamic forces produced by those surfaces. Thus, an understanding of propeller-wing interactions is essential for the design and analysis of many aircraft.

There are multiple existing methods for analyzing the propeller-wing interactions. High order methods, such as wind tunnel testing or computational fluid dynamics, provide very accurate results but come at a high cost in computation or labor. Low-order methods provide results with good accuracy at a significantly lower cost. Thus, it is desirable to use low-order methods for initial design and utilize higher order methods closer to the end of the design phase.

Current low-order models for propeller-wing interactions give reasonable results, but have shortcomings in either computational cost or accuracy. In an effort to improve on these existing models, an improved low-order model for propeller-wing interactions is proposed. This improved model utilizes several aerodynamic models such as blade element theory and lifting line theory as well as a novel turbulent prop wash model. The final model is shown to provide more accurate results using efficient numerical methods.
ACKNOWLEDGMENTS

There are many individuals to whom I owe thanks for their support in my journey. First, I would like to thank Dr. Doug Hunsaker. His mentoring and support allowed me to pursue something I am truly passionate about and learn much about aerodynamics and myself in the process.

Second, I would like to thank my parents. Their encouragement and support throughout my entire life helped me to shine both academically and personally. I’d also like to thank them for making graduate school mandatory through their examples.

Finally, I would like to thank my wife, family, colleagues, and friends for their support and enthusiasm. I have been surrounded by wonderful people in my life and owe much of who I am and what I have accomplished to them.
## CONTENTS

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>iii</td>
</tr>
<tr>
<td>PUBLIC ABSTRACT</td>
<td>iv</td>
</tr>
<tr>
<td>ACKNOWLEDGMENTS</td>
<td>v</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>vii</td>
</tr>
<tr>
<td>CHAPTER</td>
<td></td>
</tr>
<tr>
<td>1. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>2. NUMERICAL BLADE ELEMENT MODEL</td>
<td>9</td>
</tr>
<tr>
<td>3. TURBULENT PROP WASH MODEL</td>
<td>33</td>
</tr>
<tr>
<td>4. NUMERICAL LIFTING LINE MODEL</td>
<td>73</td>
</tr>
<tr>
<td>5. CONCLUSION</td>
<td>87</td>
</tr>
<tr>
<td>REFERENCES</td>
<td>91</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>12</td>
</tr>
<tr>
<td>2.2</td>
<td>13</td>
</tr>
<tr>
<td>2.3</td>
<td>15</td>
</tr>
<tr>
<td>2.4</td>
<td>18</td>
</tr>
<tr>
<td>2.5</td>
<td>23</td>
</tr>
<tr>
<td>2.6</td>
<td>24</td>
</tr>
<tr>
<td>2.7</td>
<td>26</td>
</tr>
<tr>
<td>2.8</td>
<td>28</td>
</tr>
<tr>
<td>2.9</td>
<td>29</td>
</tr>
<tr>
<td>2.10</td>
<td>30</td>
</tr>
<tr>
<td>2.11</td>
<td>30</td>
</tr>
<tr>
<td>2.12</td>
<td>31</td>
</tr>
<tr>
<td>2.13</td>
<td>32</td>
</tr>
<tr>
<td>3.1</td>
<td>38</td>
</tr>
<tr>
<td>3.2</td>
<td>40</td>
</tr>
<tr>
<td>3.3</td>
<td>43</td>
</tr>
<tr>
<td>3.4</td>
<td>50</td>
</tr>
<tr>
<td>3.5</td>
<td>54</td>
</tr>
<tr>
<td>3.6</td>
<td>56</td>
</tr>
<tr>
<td>3.7</td>
<td>59</td>
</tr>
<tr>
<td>Section</td>
<td>Description</td>
</tr>
<tr>
<td>---------</td>
<td>-------------</td>
</tr>
<tr>
<td>3.8</td>
<td>Numerical model of development of prop wash of GWS 5x4.3 propeller</td>
</tr>
<tr>
<td>3.9</td>
<td>Numerical prop wash vs. experimental measurements for GWS 5x4.3 propeller at J=0</td>
</tr>
<tr>
<td>3.10</td>
<td>Numerical prop wash vs. experimental measurements for GWS 5x4.3 propeller at J=0.52</td>
</tr>
<tr>
<td>3.11</td>
<td>Numerical prop wash vs. experimental measurements for APC 4.2x2 propeller at J=0</td>
</tr>
<tr>
<td>3.12</td>
<td>Numerical prop wash vs. experimental measurements for DA4002 5x3.75 propeller at J=0</td>
</tr>
<tr>
<td>3.13</td>
<td>Numerical prop wash vs. experimental measurements for DA4002 9x6.5 propeller at J=0</td>
</tr>
<tr>
<td>3.14</td>
<td>Numerical prop wash vs. experimental measurements for DA4002 9x6.5 propeller at J=0.64</td>
</tr>
<tr>
<td>3.15</td>
<td>Inviscid vs. turbulent prop wash model results for GWS 5x4.3 propeller at J=0</td>
</tr>
<tr>
<td>3.16</td>
<td>Axial and angular momentum as function of radial nodes</td>
</tr>
<tr>
<td>3.17</td>
<td>Percent error of momentums as function of radial nodes</td>
</tr>
<tr>
<td>4.1</td>
<td>Horseshoe vortices used in numerical lifting line method</td>
</tr>
<tr>
<td>4.2</td>
<td>Lift distribution across wing semispan with varied application of prop wash reduction factors</td>
</tr>
<tr>
<td>4.3</td>
<td>Lift distribution from Epema vs. NLLM with ARF=1 &amp; SRF=0.6</td>
</tr>
<tr>
<td>4.4</td>
<td>Lift distribution from Veldhuis vs. NLLM with ARF=1 &amp; SRF=0.6</td>
</tr>
<tr>
<td>4.5</td>
<td>Lift distribution from Stuper vs. NLLM with ARF=1 &amp; SRF=0.6</td>
</tr>
<tr>
<td>4.6</td>
<td>Lift and rolling moment coefficients as function of spanwise nodes</td>
</tr>
<tr>
<td>4.7</td>
<td>Percent error in $C_L$ &amp; $C_t$ as function of spanwise nodes</td>
</tr>
</tbody>
</table>
CHAPTER 1
INTRODUCTION

1.1 Research Motivation

Propellers and the prop wash they create can have a strong effect on the aerodynamics of many aircraft. As the prop wash from a propeller flows over the lifting surfaces of an aircraft it causes an increase in dynamic pressure and changes the local angle of attack. This can result in many different effects including an altered lift distribution, increased control surface authority, and changes in the aircraft stability\(^1\). With a sound understanding of the effects of the prop wash on the aerodynamics of an aircraft, propeller and lifting-surface characteristics can be optimized to produce a range of benefits including increased lift and reduced drag\(^2\). Therefore, a computationally efficient and accurate model of the effects of propeller wash on lifting surfaces could be extremely beneficial to create more efficient aircraft.

1.2 Literature Review

The concept of computationally modeling the effects of prop wash on lifting surfaces is not new or unprecedented. Because of the significant effect that propellers can have on the aerodynamics of many aircraft, multiple methods have been proposed and employed for modeling the effects of said propellers. These methods have been employed with different levels of computational efficiency and varying degrees of accuracy. Presented here is an overview of some of the more noteworthy methods.

Rather extensive work has been done by various researchers from the Delft University of Technology in developing methods for modeling the effect of prop wash on lifting surfaces. L. L. M. Veldhuis is one of the more notable of these researchers\(^1,2\). In his
research, he developed a number of numerical methods for modeling the interactions between propellers and lifting surfaces. These methods ranged from a simpler Vortex Lattice Method (VLM) to a more complex Navier-Stokes (NS) CFD solver. Through his research, he noted that the computationally efficient VLM code provided quite accurate results, but only when a swirl recovery factor was applied to reduce the magnitude of the change in local angles of attack due to the tangential velocity in the propeller slipstream. Additionally, he noted that although the VLM code gave accurate results, once details of the flow field are needed, the NS solver became necessary as the VLM code did not provide this information.

H. K. Epema, another researcher from the Delft University of Technology, expanded on the work done by Veldhuis by developing two codes for analyzing the propeller-wing interaction, one using a Lifting Line (LL) method and the other using VLM. Both of these methods were combined with a propeller analysis tool (XROTOR) and incorporated the swirl recovery factor proposed by Veldhuis. These methods were then compared to experimental data gathered from wind-tunnel testing. His results found that while both the LL and VLM methods showed the expected effects of the propeller on the wing, there was some discrepancy in the magnitude of those effects. Even with the swirl recovery factor, his methods tended to over predict the lift produced by the wing in the slipstream of the propeller. He found that more accurate results were found by removing the axial component of the prop wash velocity in the calculation of the forces produced by the wing, but the reason for this is not well understood. Therefore, he states that these methods can be used for optimization as long as the results are carefully interpreted.

Another notable method for modeling propeller-wing interactions was developed
by Douglas Hunsaker at Brigham Young University. His approach combined a modern adaptation of Prandtl’s Lifting-Line theory, as set forth by Phillips, with a blade-element-propeller model to calculate the influence of the propeller on the lifting surfaces. To do this, the induced velocities in the slipstream of the propeller were computed at the control points of the lifting-line code using an approach suggested by Stone. These induced velocities were then added to the local velocity vectors of those immersed control points and the lifting-line vortex system was solved. Using this approach, Hunsaker noted that while the effects of the prop matched qualitatively with what was expected, the combined model tended to over predict the amount of influence that the prop wash had on lifting surfaces. However, even with this shortcoming, his model was successfully integrated into a 6-DOF simulator showing its value as an efficient and sufficiently accurate aerodynamic analysis tool.

In addition to the models above that compute the complete interaction between the propeller and the wing, there are multiple existing methods for modeling just the development of the prop wash. As the prop wash development is an integral part of any complete model for prop-wing interactions, several of these methods are presented here.

As previously stated in the description of Hunsaker’s work, one notable method for modeling the prop wash development was created by Stone. In this method, the propeller slipstream is modeled as a series of concentric stream tubes whose inner and outer radii align with the blade elements of the propeller model. The induced axial velocity in the slipstream is scaled by a slipstream development factor, as given by McCormick to show the acceleration of the slipstream due to the helical vortex system trailing the propeller. Conservation of mass and conservation of angular momentum are then applied to each of
these stream tubes to determine the amount of contraction of the propeller slipstream as it develops downstream. The main shortcoming of this method is that it does not show the effects of turbulent mixing within the slipstream or with the freestream. However, despite this shortcoming, it remains valuable as a simple and robust approach to prop wash modeling that can be further improved.

Another prop wash model of note was developed by Khan. What sets Khan’s model apart is that it attempts to show the effects of turbulent mixing on the prop wash. This is done by dividing the prop wash into several regions: the near-field region and the far-field region. In the near-field region, the velocity in the propeller slipstream is modeled as a uniform, average profile that accelerates and contracts, as expected in an inviscid slipstream model. Then, a short distance downstream from the propeller, the slipstream transitions to the far-field region where it is modeled using equations originally developed for the boat screws that show how the wake expands and develops due to turbulent mixing. These equations are modified to reflect the viscosity of air instead of water and are then applied to the propeller slipstream. While this approach does reflect the diffusion of the prop wash due to turbulent mixing, it does have several shortcomings. Namely it does not show the radial variations in the induced velocity in the near-field region and, more importantly, it only works for static thrust cases, where the freestream velocity is 0. However, despite these shortcomings, the approach is novel and could be expanded on to improve its performance.

1.3 Proposed Low-Order Model

As can be seen in the literature presented above, low-order methods can be employed to efficiently provide results with reasonable accuracy showing the effect of
propellers on lifting surfaces. Employing these low-order methods is desirable because they are very computationally efficient and easy to use, and therefore lend themselves well to optimization and first-stage, conceptual design. Therefore, a new low-order method is proposed that combines a blade element model, a novel turbulent prop wash model, and a modern numerical lifting line model to provide results of increased accuracy while maintaining high computational efficiency.

Blade Element Theory (BET) is a common method for calculating the aerodynamics of propellers and rotors. In this method, a propeller blade is subdivided along its span into a number of cross-sectional elements. The aerodynamics of these sections are then analyzed individually and the total thrust and torque for the propeller as a whole are determined by summing up the forces of each of these blade elements. In order to do this, several assumptions about the trailing vortex system and the velocities it induces are made. However, in spite of these assumptions, BET consistently gives accurate results for a large range of propeller geometries and operating conditions. Additionally, because of the analysis of the individual sections, BET also provides the induced axial and tangential velocities along the span of the blade. These velocities can then be used to determine the development of the propeller wash.

From the literature presented above, one clear shortcoming in the majority of existing prop wash models is that the effects of turbulent mixing are completely neglected. Most models simply assume that the prop wash is purely inviscid and therefore only model the expected contraction and acceleration of an inviscid propeller slipstream. This approach provides reasonably accurate results when analyzing the prop wash of propellers at high advance ratios. However, when analyzing cases containing either a propeller in a
static thrust condition or a lifting surface far downstream from the propeller, such as an empennage, the effects of turbulent mixing become more apparent and neglecting them can produce less accurate results.

To remedy this shortcoming, a new method for modeling prop wash development is proposed that uses observations from the development of turbulent jets to apply a number of corrections to an inviscid propeller slipstream. Although the effects of turbulent mixing on propeller slipstreams are not well understood, turbulent jets and characteristics of their development have been extensively studied and modeled with very good accuracy for years. Qualitative observation of experimental measurements of propeller slipstreams shows that propeller slipstreams exhibit many of the same characteristics in their development as turbulent jets. Thus, by applying an understanding of the development of turbulent jets, corrections can be made to the inviscid propeller slipstream to show how the slipstream develops under the effects of turbulent mixing. This new method for propeller slipstream modelling provides more accurate results over a much broader range of configurations and operating conditions.

Once the propeller slipstream development has been accurately modeled, its effect on lifting surfaces can be calculated. To do this, a dimensional adaptation of the modern numerical lifting line method proposed by Phillips is employed. This numerical lifting line method simulates lifting surfaces as a system of horseshoe vortices and has been shown to give accurate results at a significantly lower computational cost than similar low-order methods. To determine the effects of the prop wash on a lifting surface, the local velocity vectors of the control points immersed in the prop wash are altered to account for the additional velocity induced by the prop wash itself. This alters the local angle of attack and
dynamic pressure and is reflected in an altered forces and moments distribution across the span of the lifting surface. A number of reduction factors are also applied to the applied prop wash velocities to improve the agreement with experimental data.

The results produced by each component of the implemented model are compared to experimental results and their accuracy is evaluated. Research conducted by the University of Illinois provides a database of propeller performance measurements and prop wash velocity measurements to which the results of the propeller blade element model and the turbulent prop wash model can be compared\textsuperscript{10}. Research conducted by Veldhuis, Epema, and Stuper also provides a number of experiments of the effects of propellers on wings against which the complete propeller-wing model can be compared and evaluated\textsuperscript{1,2,17}. The results of each component of the model and of the complete model are compared to experimental results qualitatively and quantitatively to ensure good agreement. A number of semi-empirical correction factors are added to various aspects of the model to improve the level of agreement of the results when compared to the range of experimental data available.

The principle benefit of the proposed method is that it provides both improved speed and accuracy when compared to similar low-order methods. This makes it particularly useful for multiple applications including conceptual design and optimization. Using this method, a large range of configurations and operating conditions can be quickly analyzed to establish a design space. Optimal points on this design space can then be further analyzed using methods such as CFD and wind tunnel testing to create a fully optimized design. Additionally, because both the propellers and lifting surfaces can be fully defined by a small number of simple parameters, this method lends itself very well to quick,
intuitive design and analysis. Ultimately, this method will be incorporated into the newest
version of MachUp, a free, open-source aerodynamic analysis tool developed by the USU
Aerolab that is also an easy-to-use online tool available to the general public.
CHAPTER 2
NUMERICAL BLADE ELEMENT MODEL

2.1 Nomenclature

\( c_b \) = blade section chord length

\( \tilde{C}_D \) = blade section drag coefficient

\( \tilde{C}_{D0} \) = first drag polar coefficient, Eq. (2.28)

\( \tilde{C}_{DL} \) = second drag polar coefficient, Eq. (2.28)

\( \tilde{C}_{DL2} \) = third drag polar coefficient, Eq. (2.28)

\( \tilde{C}_L \) = blade section lift coefficient

\( \tilde{C}_{L,\alpha} \) = blade section lift slope coefficient

\( \tilde{C}_{L,\text{max}} \) = max lift coefficient at stall

\( \tilde{D} \) = blade section drag force

\( d_p \) = propeller diameter

\( f \) = Prandtl’s tip loss factor, Eq. (2.19)

\( \tilde{F}_{\theta} \) = blade section circumferential force

\( k \) = number of propeller blades

\( \tilde{L} \) = blade section lift force

\( \tilde{T} \) = blade section thrust force

\( r \) = radial distance from propeller axis

\( V_b \) = total local velocity

\( V_i \) = induced velocity
\( V_t \) = tangential induced velocity  
\( V_{xi} \) = axial induced velocity  
\( V_\infty \) = freestream velocity  
\( \alpha \) = aerodynamic angle of attack  
\( \alpha_{\text{stall}} \) = aerodynamic angle of attack at which stall occurs  
\( \Delta \alpha \) = angle of attack shift for rotational stall delay, Eq. (2.39)  
\( \beta \) = aerodynamic pitch angle  
\( \beta_t \) = aerodynamic pitch angle at blade tip  
\( \varepsilon_b \) = downwash angle, Eq. (2.2)  
\( \varepsilon_i \) = induced angle, Eq. (2.3)  
\( \varepsilon_\infty \) = advance angle, Eq. (2.1)  
\( \kappa \) = Goldstein’s Kappa factor  
\( \Gamma \) = blade section circulation  
\( \lambda \) = aerodynamic pitch  
\( \lambda_c \) = aerodynamic pitch  
\( \rho \) = freestream density  
\( \omega \) = propeller rotation rate

### 2.2 Background

There are multiple methods for numerically determining the aerodynamics and performance of propellers. These methods range from low- to high-order and each has its own level of accuracy and computational cost. Propeller Momentum Theory is an example of low-order method, as it assumes that the flow through a propeller is one-dimensional and it neglects rotation of the propeller slipstream\(^9\). This results in an idealized, optimistic
prediction of propeller performance. A method developed by Montgomery\textsuperscript{18} is an example of a high-order method. It applies Phillip’s numerical lifting-line method to model the aerodynamics of propellers\textsuperscript{18}. Unfortunately, this method has a significantly increased computational cost when compared to Blade Element Theory while providing only a marginal increase in accuracy and utility\textsuperscript{18}. Blade Element Theory provides a good balance of accuracy and computational efficiency and is therefore employed as the first component of the proposed low-order model.

2.3 Blade Element Theory

The derivation of Blade Element Theory as presented here follows the derivation by Phillips\textsuperscript{8}. In order to predict the performance of a propeller, it is important to understand the aerodynamics acting on the individual blades. For this purpose, the aerodynamics acting on a cross section of a single propeller blade as shown in Figure 2.1 are considered.

In this case, the propeller is rotating with angular velocity $\omega$ and moving forward through the air with velocity $V_\infty$. Additionally, its axis is considered to be aligned with the direction of forward motion. A correction can be added to account for off-axis forces and moments produced by an angle of incidence between the propeller axis and the direction of motion. This correction is derived by Phillips\textsuperscript{19}, and is included in the complete propeller-wing model, but will not be explained in this work.

There are a number of aerodynamic angles that are important to this analysis and are shown in Figure 2.1. The angle that the zero-lift line of the cross section makes with the propeller axis of rotation is known as the aerodynamic pitch angle, $\beta$. The total downwash angle, $\varepsilon_b$, is made of two components: the advance angle, $\varepsilon_\infty$, and the induced angle, $\varepsilon_i$. The advance angle is created by the forward motion of the propeller and the
induced angle is caused by the vortex shedding from the tips of the propeller blades. The angle between the local free stream velocity vector, $V_b$, and the zero-lift line of the cross section is known as the aerodynamic angle of attack, $\alpha$.

![Fig. 2.1 Cross-sectional element of blade with local velocities and angles.](image)

With the known rotation rate of the propeller and forward velocity, the advance angle can be determined from the geometry shown in Figure 2 as

$$\epsilon_a = \arctan \left( \frac{V_\infty}{\omega r} \right) \quad (2.1)$$

Likewise, if the induced velocity, $V_i$, is separated into a tangential component, $V_{\theta i}$, and an axial component, $V_{xi}$, as shown in Figure 2.2, the total downwash angle can be determined from the same geometry as

$$\epsilon_b = \arctan \left( \frac{V_\infty + V_{xi}}{\omega r - V_{\theta i}} \right) \quad (2.2)$$

The induced angle is defined as the total downwash angle minus the advance angle as

$$\epsilon_i = \epsilon_b - \epsilon_a \quad (2.3)$$

Similar to a wing, downwash on the propeller blade section tilts the lift and drag vectors back as shown in Figure 2.2.
Fig. 2.2 Cross sectional blade element showing induced velocity components and force vectors.

This tilting of the lift and drag vectors ultimately add to the torque required to rotate the propeller and decreases the total thrust developed by the propeller. From the geometry shown in Figure 2.2, the section thrust and circumferential forces are related to the section lift and drag as

\[
\tilde{T} = \tilde{L} \cos(\varepsilon_b) - \tilde{D} \sin(\varepsilon_b) \\
- \tilde{F}_\theta = -\tilde{L} \sin(\varepsilon_b) - \tilde{D} \cos(\varepsilon_b)
\]  

(2.4)

(2.5)

The section lift and drag forces are determined from the definition of the section coefficients as

\[
\tilde{L} = \tilde{C}_L \frac{1}{2} \rho V_b^2 c_b
\]

(2.6)

\[
\tilde{D} = \tilde{C}_D \frac{1}{2} \rho V_b^2 c_b
\]

(2.7)

The local velocity magnitude, \(V_b\), is determined from the geometry as shown in Figure 2.2 as

\[
V_b^2 = \omega^2 r^2 \left[ 1 - \frac{V_{ao}}{\omega r} + \left( \frac{V_{ao}}{\omega r} + \frac{V_{ao}}{\omega r} \right) \right]
\]

(2.8)

Combining Eq. (2.8) with Eqs. (2.6-2.7), the section thrust and circumferential
forces are

\[ \vec{T} = \frac{1}{2} \rho \omega^2 r^2 \xi_b \left[ \left(1 - \frac{V_\theta}{\omega r} \right) + \left( \frac{V_x}{\omega r} + \frac{V_{xi}}{\omega r} \right) \right] \left( \bar{C}_L \cos(\epsilon_b) - \bar{C}_D \sin(\epsilon_b) \right) \] (2.9)

\[ \vec{F}_\theta = -\frac{1}{2} \rho \omega^2 r^2 \xi_b \left[ \left(1 - \frac{V_\theta}{\omega r} \right) + \left( \frac{V_x}{\omega r} + \frac{V_{xi}}{\omega r} \right) \right] \left( \bar{C}_L \sin(\epsilon_b) + \bar{C}_D \cos(\epsilon_b) \right) \] (2.10)

To determine the thrust developed for the whole propeller per radial distance, the section thrust is multiplied by the number of blades, \( k \), so that

\[ \frac{dT}{dr} = k\vec{T} = \frac{k}{2} \rho \omega^2 r^2 \xi_b \left[ \left(1 - \frac{V_\theta}{\omega r} \right) + \left( \frac{V_x}{\omega r} + \frac{V_{xi}}{\omega r} \right) \right] \left( \bar{C}_L \cos(\epsilon_b) - \bar{C}_D \sin(\epsilon_b) \right) \] (2.11)

Likewise, the torque developed per radial distance for the full prop is equal to the section circumferential force multiplied by the number of blades and the radial distance to the section, so that

\[ \frac{d\ell}{dr} = -kr\vec{F}_\theta = \frac{k}{2} \rho \omega^2 r^3 \xi_b \left[ \left(1 - \frac{V_\theta}{\omega r} \right) + \left( \frac{V_x}{\omega r} + \frac{V_{xi}}{\omega r} \right) \right] \left( \bar{C}_L \sin(\epsilon_b) + \bar{C}_D \cos(\epsilon_b) \right) \] (2.12)

These equations are then integrated over the radius of the propeller to determine the total thrust and torque produced by the propeller. However, in Eq. (2.9) and Eq. (2.10) there are four unknowns: \( \vec{T}, \vec{F}_\theta, V_{xi}, \) and \( V_{\theta i} \). All other values are known values of the propeller geometry or operating conditions. With four unknowns and only two equations, it is not yet possible to solve for the total thrust and torque. Thus, additional expressions must be found to determine the values of the induced velocity components. In order to do this, several simplifying assumptions must be made.

These assumptions come from an understanding of the vortex lifting law and the vorticity shed from the tips of the propeller blades. Similar to a wing, lift can only be produced with the simultaneous creation of vorticity or circulation. As with a wing, the lift
for a section of a propeller blade is related to the local circulation, $\Gamma$, by the following expression

$$\tilde{L} = \rho V_b \Gamma$$

(2.13)

Similar to a wing, the lift must go to zero at the tips of the propeller blades because the pressure difference between the upper and lower surfaces of the propeller blade cannot be supported. This means that the circulation at the tip of the propeller is shed to create vortices. It is this shed vorticity that produces the induced angle on the propeller. Due to the symmetry of the propeller, each blade receives as much downwash as it does upwash from the bound vorticity of the preceding and following blades respectively. Therefore, the induced angle is a product of the shed vorticity only and not the bound vorticity\textsuperscript{20}.

Because the propeller blades are simultaneously rotating and moving forward through the air, the shed vortices do no follow a straight or even circular path trailing the propeller blade tips. Instead, they follow a roughly helical path trailing behind the propeller. Figure 2.3 shows water vapor condensing in the core of the vortices shed from the propellers of a C-130 Hercules. This is a useful visualization and helps establish an understanding of how the shed vortices propagate downstream.

Fig. 2.3 Tip vortices shed from propellers of C-130 Hercules (U.S. Air Force).
Calculating the induced velocity in the plane of the propeller from this entire helical vortex system is quite challenging. One method for doing so is known as Goldstein’s vortex theory. This theory uses two simplifying assumptions. First, it is assumed that the trailing vortices follow a helical path of constant pitch. Second, it is assumed that the induced velocity, $V_i$, is normal to the resultant velocity, $V_b$. It has been shown that these assumptions are both satisfied in the slipstream of an optimum propeller. This is known as the Betz condition$^{20}$. For the case of a non-optimum propeller of arbitrary geometry, McCormick states that “studies have been performed that support normality at the plane of the propeller” and he has also shown that Goldstein’s vortex theory gives reasonable results$^{21}$.

From the normality hypothesis and the geometry as previously shown in Figure 2.2, the resultant velocity is related to the downwash angle as

$$V_b = \left( \omega^2 r^2 + V_\infty^2 \right) \cos(\varepsilon_i) = \omega r \cos \left( \varepsilon_i \right) \cos \left( \varepsilon_\infty \right) \tag{2.14}$$

Therefore, the induced velocity is defined as

$$V_i = \omega r \frac{\sin(\varepsilon_i)}{\cos(\varepsilon_\infty)} \tag{2.15}$$

The induced velocity is then broken into its axial and tangential components as

$$V_{\alpha i} = V_i \cos(\varepsilon_\infty + \varepsilon_i) = \omega r \frac{\sin(\varepsilon_i)}{\cos(\varepsilon_\infty)} \cos(\varepsilon_\infty + \varepsilon_i) \tag{2.16}$$

$$V_{\theta i} = V_i \sin(\varepsilon_\infty + \varepsilon_i) = \omega r \frac{\sin(\varepsilon_i)}{\cos(\varepsilon_\infty)} \sin(\varepsilon_\infty + \varepsilon_i) \tag{2.17}$$

Thus, the components of the induced velocity become functions of the induced angle, $\varepsilon_i$. This angle can be determined from the assumption of a helical trailing vortex system of constant pitch. Goldstein’s vortex theory predicts that the tangential component of the induced velocity, $V_{\theta i}$, is related to the local section circulation, $\Gamma$, by the following
expression

\[ k\Gamma = 4\pi \kappa V_{\theta} \]  \hspace{1cm} (2.18)

Where \( \kappa \) is known as Goldstein’s kappa factor. This factor is available in graphical form, but a closed form has never been presented. However, this factor can be closely approximated by use of Prandtl’s tip loss factor in place of \( \kappa^2 \). Prandtl’s tip loss factor is given as

\[ f = \frac{2}{\pi} \cos^{-1}\left(\exp\left(-\frac{k\left(1 - \frac{2r}{d_p}\right)}{2\sin(\beta_t)}\right)\right) \]  \hspace{1cm} (2.19)

where \( \beta_t \) is the aerodynamic pitch angle at the blade tip. Combining Eq. (2.6) and Eq. (2.13), the local circulation is related to the local lift coefficient as

\[ \Gamma = \frac{1}{2} V_b c_b \tilde{C}_L = \frac{1}{2} \omega c_b \tilde{C}_L \frac{\cos(\varepsilon_i)}{\cos(\varepsilon_{\infty})} \]  \hspace{1cm} (2.20)

Finally, combining Eqs. (2.17-2.20), the following expression is derived

\[ \frac{k c_b}{16r} \cos^{-1}\left(\exp\left(-\frac{k\left(1 - \frac{2r}{d_p}\right)}{2\sin(\beta_t)}\right)\right) \tan(\varepsilon_i) \sin(\varepsilon_{\infty} + \varepsilon_i) = 0 \]  \hspace{1cm} (2.21)

In this equation, the only unknown value is the induced angle, \( \varepsilon_i \). Therefore, this equation can be solved numerically to determine the induced angle. This can be done using a variety of root-finding methods. However, it should be noted that finding the correct root of this function can be tricky. Figure 2.4 shows the value of the right hand side of equation 2.21 as a function of induced angle for an arbitrary propeller blade section.
As can be seen in Figure 2.4, there are two values of the induced angle that satisfy Eq. (2.21). This can cause difficulties when attempting to find the correct value of the induced angle using a numerical solver, unless several requirements are imposed to arrive at the correct solution.

First, the induced angle must fall within a range of -90 to 90 degrees. While this requirement seems obvious, there are times when use of the secant method will result in an induced angle on the order of magnitude of thousands of degrees. This is not physically feasible, so a limit is placed on the numerical method that only allows it to make guesses within a range of -90 to 90 degrees.

Second, the sign of the induced angle must match the sign of the difference between the aerodynamic pitch angle and the advance angle, $\beta - \xi^\infty$. This requirement helps to distinguish between the two possible roots and makes sense if Eq. (2.21) is analyzed. Consider a case where the advance angle is greater than the aerodynamic pitch angle. For this case, the airfoil section will have a negative angle of attack, resulting in a negative
value for the section lift coefficient, \( \tilde{C}_L \). This requires that the second term in Eq. (2.21) also be negative so that the right hand side equals zero. Analysis of this second term shows that the only case that can satisfy this condition is when the \( \tan \epsilon_i \) is negative, meaning that \( \epsilon_i \) is negative. Similar analysis also shows that for the case when the advance angle is less than the aerodynamic pitch angle, the induced angle must be positive. Therefore, the sign of \( \epsilon_i \) must match the sign of \( \beta - \epsilon_\infty \).

Imposing these two requirements on the numerical method used to solve Eq. (2.21) ensures that the correct solution will always be found within the range of normal operation.

Once the induced angle is known, it can be inserted into the following equations, which were obtained by inserting Eq. (2.14) into Eq. (2.11) and Eq. (2.12) and integrating, to determine the total thrust and torque developed by the propeller.

\[
T = \frac{k}{2} \rho \omega^2 \int_{r_o}^{r_i} r^2 c_h \frac{\cos^2(\epsilon_i)}{\cos^2(\epsilon_\infty)} \left( \tilde{C}_L \cos(\epsilon_\infty + \epsilon_i) - \tilde{C}_D \sin(\epsilon_\infty + \epsilon_i) \right) dr
\]  
\[\text{(2.22)}\]

\[
\ell = \frac{k}{2} \rho \omega^2 \int_{r_o}^{r_i} r^3 c_h \frac{\cos^2(\epsilon_i)}{\cos^2(\epsilon_\infty)} \left( \tilde{C}_L \sin(\epsilon_\infty + \epsilon_i) + \tilde{C}_D \cos(\epsilon_\infty + \epsilon_i) \right) dr
\]  
\[\text{(2.23)}\]

### 2.4 Aerodynamics of 2-D airfoil sections

In the derivation of the numerical blade element model, one of the key factors is an accurate representation of the lift and drag coefficients of the propeller blade elements at varying angles of attack. As shown in Eq. (2.21), the lift coefficient has a direct influence on the induced angle. Additionally, if the lift and drag coefficients are inaccurate, the values of thrust and torque will be inaccurate as well, as shown in Eqs. (2.22-2.23). Therefore, for the numerical blade element model to produce accurate results, the propeller blade element’s airfoil properties must be well understood.
2.4.1 Pre-stall Airfoil Properties

To define the 2D section airfoil properties, there are three variables that define the
lift coefficient slope and three variables that define the drag polar. For the lift coefficient,
these three variables are the zero-lift angle of attack, the lift slope, and the max lift
coefficient.

Because all angles in the blade element model are relative to the zero-lift line of the
airfoil, the zero-lift angle of attack only comes into play in the initial calculations of the
aerodynamic pitch angle. It should be noted that most tabulated propeller data has the pitch
of the propeller measured relative to the section chord line or to a flat lower surface of the
airfoil section. Therefore, it is necessary to convert the manufacturer specified chord-line
pitch, \( \lambda_c \), to an aerodynamic pitch, \( \lambda \), in order for it to be used in the blade element model.
The relation between chord-line pitch and aerodynamic pitch is given as\(^{20}\)

\[
\lambda(r) = 2\pi r \frac{\dot{\lambda}_c - 2\pi r \tan(a_{\lambda,0})}{2\pi r + \dot{\lambda}_c \tan(a_{\lambda,0})}
\]  

(2.24)

The aerodynamic pitch angle, \( \beta \), is then determined from the aerodynamic pitch as

\[
\beta(r) = \tan^{-1}(\frac{\lambda(r)}{2\pi r})
\]

(2.25)

Since the angle of attack in the blade element model is measured relative to the zero-lift
line of the airfoil, the lift coefficient is simply defined as

\[
\tilde{C}_L = \tilde{C}_{L,0} \alpha
\]

(2.26)

The maximum lift coefficient, \( \tilde{C}_{L,\text{max}} \), is defined as the maximum lift coefficient achieved
before stall. This max lift coefficient is used to calculate the angle of attack where the
airfoil stalls as
\[ \alpha_{\text{stall}} = \frac{\tilde{C}_{L,\text{max}}}{\tilde{C}_{L,\alpha}} \] (2.27)

For angles of attack below \( \alpha_{\text{stall}} \) the simple, linear calculation of the lift coefficient as shown in Eq. (2.26) is used. For angles of attack beyond \( \alpha_{\text{stall}} \) a post-stall model must be implemented. This post-stall model will be discussed later.

It is also important to note that for cambered airfoils, \( \tilde{C}_{L,\text{max}} \) is typically different for negative angles of attack than it is for positive angles. Because of the camber, an airfoil will typically stall at a lower angle of attack when inverted. This behavior should be taken into account to provide a more accurate representation of airfoil properties.

To determine the drag, the drag coefficient is characterized as a quadratic function of \( \tilde{C}_{L} \) as shown below

\[ \tilde{C}_D = \tilde{C}_{D0} + \tilde{C}_{DL} \tilde{C}_L + \tilde{C}_{DL2} \tilde{C}_L^2 \] (2.28)

Similar to the lift coefficient, this formulation is only valid at angles of attack below stall. For angles of attack past stall, another model must be implemented. All of these airfoil coefficients can be determined by fitting curves to aerodynamic data obtained from software such as XFOIL or wind-tunnel tests.

2.4.2 Post Stall Airfoil Properties

There are various methods for extrapolating airfoil coefficients for angles of attack past stall. One method that is commonly used is the Viterna method\(^{22} \). For angles of attack from stall to 90°, the lift and drag coefficients are given as

\[ \tilde{C}_L = A_1 \sin(2\alpha) + A_2 \frac{\cos^2(\alpha)}{\sin(\alpha)} \] (2.29)

\[ \tilde{C}_D = B_1 \sin^2(\alpha) + B_2 \cos(\alpha) \] (2.30)
where

\[ \tilde{C}_{D,90^\circ} \approx 1.11 + 0.018AR \]  \hspace{1cm} (2.31)

\[ A_1 = \frac{\tilde{C}_{D,90^\circ}}{2} \]  \hspace{1cm} (2.32)

\[ B_1 = \tilde{C}_{D,90^\circ} \]  \hspace{1cm} (2.33)

\[ A_2 = \left( \tilde{C}_{L,max} - \tilde{C}_{D,90^\circ}\sin(\alpha_{stall})\cos(\alpha_{stall}) \right) \left( \frac{\sin(\alpha_{stall})}{\cos^2(\alpha_{stall})} \right) \]  \hspace{1cm} (2.34)

\[ B_2 = \frac{\tilde{C}_{D,max} - \tilde{C}_{D,90^\circ}\sin^2(\alpha_{stall})}{\cos(\alpha_{stall})} \]  \hspace{1cm} (2.35)

In Eq. (2.31), \( AR \) is the aspect ratio of the propeller blade. This value is important because the finite length of the blade affects the flat plate assumption that is the basis of the Viterna method\(^{22}\). Additionally, \( \tilde{C}_{D,max} \) is the value of the drag coefficient that corresponds to the max lift coefficient, \( \tilde{C}_{L,max} \), as previously used in Eq. (2.27).

Using this method, the lift and drag coefficients can be extrapolated for angles of attack from stall to 90\(^\circ\). However, one shortcoming of this method is that it produces a discontinuity in the lift slope at \( \alpha_{stall} \). This discontinuity can prove troublesome for numerical solvers attempting to solve Eq. (2.21), making it harder to converge to a solution. Therefore, an additional function is used to blend the pre- and post-stall models so that there is no discontinuity in the lift slope at stall. This blend function is given as

\[ \tilde{C}_L = (1 - S)\tilde{C}_{L,\text{linear}} + S\tilde{C}_{L,\text{Viterna}} \]  \hspace{1cm} (2.36)

where \( \tilde{C}_{L,\text{linear}} \) and \( \tilde{C}_{L,\text{Viterna}} \) are given by Eq. (2.26) and Eq. (2.29) respectively and

\[ S = \tanh\left( \frac{(\alpha - \alpha_{stall})}{2} + 0.5 \right) \]  \hspace{1cm} (2.37)
It should be noted that Eq. (2.36) should only be used at angles of attack close to stall. At very large angles of attack or near zero, the values of $\tilde{C}_{L,\text{linear}}$ and $\tilde{C}_{L,\text{visc}}$ respectively can become very large and disrupt the calculation of the lift coefficient. Figure 2.5 shows a comparison of the original functions spliced together and the blended functions at angles of attack near stall. This comparison uses a lift slope of $2\pi$ and a $\tilde{C}_{L,\text{max}}$ of 1.4.

![Comparison of original and blended $C_L$ functions.](image)

**Fig. 2.5 Comparison of original and blended $C_L$ functions.**

Figure 2.6 shows the final curves for the lift and drag coefficients of a 2D airfoil section with $\tilde{C}_{L,\alpha} = 2\pi$, $\tilde{C}_{L,\text{max}} = 1.4$, $\tilde{C}_{D0} = 0.006$, $\tilde{C}_{DL} = 0$, and $\tilde{C}_{DL2} = 0.01$. While there is a discontinuity in the drag slope at $\alpha_{\text{stall}}$, there is no need to blend the pre- and post-stall drag models because the drag coefficient is not used in an iterative solver like the lift coefficient.
2.4.3 Rotational Stall-Delay Effects

One challenge with computational modeling of propellers is that performance is typically under predicted at low advance ratios. As early as 1945, Himmelskamp discovered that propeller airfoil sections performed better than would be expected based on the 2-D airfoil characteristics\textsuperscript{23}. This happens because the rotational motion of the propeller helps to delay stall to angles of attack in excess of what would normally be seen on a 2-D airfoil section.

To understand why this phenomenon occurs, consider a propeller blade element close to the hub of the propeller. At low advance ratios, this blade element sees very high angles of attack, causing it to stall. As it stalls, a region of turbulent air is trapped on top of the blade element in the separation region. This turbulent air is then pulled along with the propeller blade as the blade continues to move and rotate. As this turbulent air rotates with
the blade, it gains momentum and Coriolis forces push it radially outwards along the blade. As this air moves radially outward, it produces a suction in the boundary layer of the region it just vacated. This in turn pulls the separated boundary layer back towards the blade surface and results in a delayed stall on the propeller blade element. In reality, this whole process happens instantaneously, but it is useful to think of it as a process to fully understand what is happening.

Various models have been proposed to extend 2-D airfoil data so that propeller performance predictions match experimental results\textsuperscript{23–27}. The downfall of all of these methods is that they were all developed to match a certain set of experimental results, and each contains various variables that must be changed to match experimental results. Therefore, they are only accurately applicable in a limited range of situations where experimental results are already available. For purely conceptual design, these models can only provide a general approximation of the effects of rotational stall delay.

Several studies have been performed comparing the performance of the various models\textsuperscript{24,26}. Among these studies, there seems to be a common consensus that the model developed by Corrigans and Schillings\textsuperscript{27} most accurately reflects experimental results in a wide range of cases. Because of this, the Corrigans and Schillings model will be presented here.

For the Corrigans and Schillings model\textsuperscript{24}, the lift coefficient curve is shifted by an angle, $\Delta \alpha$, so that

$$
\tilde{C}_{L,3D}(\alpha) = \tilde{C}_{L,2D}(\alpha - \Delta \alpha) + \tilde{C}_{L,\alpha} \Delta \alpha
$$

This effectively shifts the point of stall further up the potential lift slope and lift coefficients past stall are increased by $\tilde{C}_{L,\alpha} \Delta \alpha$. This can be seen in Figure 2.7.
Fig. 2.7 Original $C_L$ compared to $C_L$ from Corrigan’s stall delay model.

This shift in angle of attack, $\Delta \alpha$, is calculated by the function

$$\Delta \alpha = \left( \frac{K \left( \frac{c}{r} \right)^n}{0.136} \right) - 1 \left( \alpha_{\text{stall}} - \alpha_{L,0} \right)$$  \hspace{1cm} (2.39)

where

$$K = \left( \frac{0.1517}{\left( \frac{c}{r} \right)} \right)^{1/1.084}$$  \hspace{1cm} (2.40)

In Eq. (2.39), Corrigans and Schillings recommend using an $n$ value between 0.8 and 2.6, varying the value to match experimental results. The value $n=1$ has shown to give good results compared to various experimental data$^{24}$.

As can be seen in Eq. (2.39) and Eq. (2.40), this stall delay model is heavily a function of $c/r$, the ratio of the local chord length to the radial position. Because of this, the
stall delay is very strong at the root, where the rotational effects are most pronounced, and tends to die off as the outer tip of the blade is approached.

Using this stall delay model improves propeller performance predictions for low advance ratios, but it does not fix it completely. Further work could be done to improve the fidelity of the rotational stall delay model, thus improving predictions of propeller performance at low advance ratios.

2.5 Results of numerical blade element model

The results of the implemented numerical blade element model are here compared to experimental measurements of propeller performance gathered by the University of Illinois. Deters, from the University of Illinois, tested a number of small-scale hobby propellers and obtained measurements of the thrust and power coefficients for a wide range of operating conditions\(^{10}\). The main focus of his work was to test the effects of low Reynolds numbers on propeller performance. One of the main effects of Reynold’s number on propeller performance is to alter the 2-D airfoil characteristics, typically by decreasing drag. Additionally, the airfoil geometry of the tested propellers was not specified. These two factors ultimately affect the accuracy of the numerical blade element model. To compensate for these factors, the coefficients of the airfoil sections in the numerical blade element model were altered until good agreement with the experimental data was achieved.

The first comparison is to a GWS 5x4.3 2-bladed hobby propeller. Figure 2.8 shows \(C_T\) and \(C_P\) measured by Deters at different RPM’s compared to the results from the numerical blade element model. The airfoil coefficients used in this model were 

\[
\alpha_{L0} = -0.0184, \tilde{C}_{L,\alpha} = 2\pi, \tilde{C}_{L,max} = 1.0, \tilde{C}_{D0} = 0.022, \tilde{C}_{DL} = 0.0045, \text{ and } \tilde{C}_{DL2} = 0.01.
\]

The dashed lines in Figure 2.8 show the results if \(\alpha_{L0} = 0\) and \(\tilde{C}_{D0} = 0.0055\). This shows
the effect that varying the airfoil parameters can have on the results. However, with the correct airfoil parameters, it can be seen that there is very good agreement over a wide range of advance ratios.

Fig. 2.8 Comparison of numerical to experimental results for GWS 5x4.3. RPM’s of experimental results are □4048, ◇6047, ◆8044, ●8078.

Figure 2.9 shows the thrust and power coefficients over a range of RPM’s for a static thrust case, meaning that the freestream velocity is zero. The experimental results show a variation in $C_T$ and $C_P$ over the range of RPM’s due to the increasing Reynold’s number. As previously mentioned, changing the Reynold’s number alters the 2D airfoil characteristics of the propeller blade, typically by decreasing drag. Thus, at low RPM’s there is a low Reynolds number resulting in higher drag and an increased power coefficient. These results are not reflected in the numerical blade element model because the airfoil characteristics were considered to be constant and independent of Reynolds number.
The second comparison is to an APC 4.2x2 Sport hobby propeller. This comparison used the following airfoil coefficients: \( \alpha_L = 0 \), \( \tilde{C}_{L,0} = 5 \), \( \tilde{C}_{L,\text{max}} = 1.0 \), \( \tilde{C}_{D0} = 0.055 \), \( \tilde{C}_{DL} = -0.0045 \), and \( \tilde{C}_{DL2} = 0.02 \). Figure 2.10 shows a comparison of \( C_T \) and \( C_P \) over a range of advance ratios. Once again, there is a good level of agreement between the experimental and numerical results over a wide range of advance ratios.

![Graph showing comparison of thrust and power coefficients](image)

**Fig. 2.9 Comparison of numerical to experimental results for GWS 5x4.3 in static thrust case.**

Figure 2.11 shows a comparison of the thrust and power coefficients for the case of static thrust. As with the previous case, the effects of the changing Reynolds number are clearly apparent in the experimental data. These effects could be modeled by calculating the airfoil coefficients of the propeller over a range of Reynolds numbers and inserting those values into the numerical blade element model. However, for the sake of simplicity, this work does not attempt to model the effects of the changing Reynolds number.
Fig. 2.10 Comparison of numerical to experimental results for APC 4.2x2. RPM’s are 6021, 9050, 12047, 12053, 15064, 15065.

Fig. 2.11 Comparison of numerical to experimental results for APC 4.2x2 in static thrust case.
From the results presented above, it can be seen that the numerical blade element model can closely model the performance of propellers. However, it must be noted that in order to do so, the geometry of the propeller, including pitch and chord distributions as well as airfoil properties, must be accurately modeled. The accuracy of this information will directly affect the accuracy of the results produced by the blade element model.

2.6 Grid resolution study of numerical blade element model

In order to test the convergence of the numerical blade element model, the effects of varying the grid resolution were tested. To do this, the thrust and power coefficients of a propeller were calculated using different levels of radial nodes. The propeller had a diameter of 1 m, three blades, an elliptic chord distribution, a root chord of 0.075 m, a pitch ratio of 0.4, and a NACA 2412 airfoil. The propeller was operating at an advance ratio of $J=0.125$. Figure 2.12 shows the values of $C_T$ and $C_P$ as a function of radial nodes.

![Fig. 2.12 $C_T$ & $C_P$ as function of radial nodes.](image-url)
Additionally, the percent error in these values was calculated using the value with the highest resolution as the reference point. These results can be seen in Figure 2.13. From this analysis, the convergence of the numerical blade element model is determined to have an order of about 2.25. This order of convergence may vary slightly depending on the propeller geometry and operating condition. Based on this analysis, it is recommended that 100 radial nodes be used in the numerical blade element model as it provides results with less than 0.05% error while maintaining good computational efficiency. Because of this, 100 radial nodes are used throughout the remainder of this work.

Fig. 2.13 Percent error in $C_T$ & $C_P$ as function of radial nodes.
CHAPTER 3
TURBULENT PROP WASH MODEL

3.1 Nomenclature

\[ b \quad = \quad \text{Gaussian half-width of mixing region} \]
\[ B \quad = \quad \text{half-width of top-hat profile} \]
\[ B^* \quad = \quad \text{dimensionless half-width of top-hat profile, Eq. (3.24)} \]
\[ c \quad = \quad \text{outer radius of self-similar tangential profile} \]
\[ D \quad = \quad \text{diameter of turbulent jet outlet} \]
\[ D_p \quad = \quad \text{diameter of propeller} \]
\[ F_{kd} \quad = \quad \text{correction factor for Slipstream Development Factor, Eq. (3.76)} \]
\[ F_w \quad = \quad \text{correction factor for radius of self-similar tangential profile, Eq. (3.49)} \]
\[ F_{um} \quad = \quad \text{correction factor for centerline velocity at } x_e, \text{ Eq. (3.50)} \]
\[ F_{wm} \quad = \quad \text{correction factor for } r_{wm}, \text{ Eq. (3.50)} \]
\[ F_\beta \quad = \quad \text{correction factor for propwash spread rate, Eq. (3.77-78)} \]
\[ k_d \quad = \quad \text{McCormick’s Slipstream Development Factor, Eq. (3.1)} \]
\[ M \quad = \quad \text{axial momentum, Eq. (3.34)} \]
\[ M_e \quad = \quad \text{excess axial momentum of coflowing jet} \]
\[ l^*_m \quad = \quad \text{momentum length scale, Eq. (3.17)} \]
\[ L \quad = \quad \text{angular momentum, Eq. (3.35)} \]
\[ r \quad = \quad \text{radial distance from prop wash centerline} \]
\[ r_{wm} \quad = \quad \text{radius of max velocity of self-similar tangential profile} \]
\[ R_p \quad = \quad \text{outer radius of propeller} \]
\[ R_{pc} \quad = \quad \text{radius of potential core of turbulent jet} \]
\( S \) = Swirl number, Eq. (3.36)
\( u \) = axial velocity
\( u_m \) = centerline velocity of fully established turbulent jet
\( u_o \) = initial uniform velocity of turbulent jet
\( u_{ss} \) = self-similar axial velocity profile of prop wash, Eq. (3.47)
\( \Delta u \) = excess axial velocity of coflowing turbulent jet
\( \Delta u_m \) = centerline excess velocity of fully established coflowing jet
\( \Delta u_{eq} \) = equivalent turbulent jet axial velocity, Eq. (3.43-44)
\( \Delta U \) = excess axial velocity of top-hat profile
\( U^* \) = dimensionless axial velocity of coflowing jet, Eq. (3.23)
\( V_ti \) = tangential induced velocity in plane of propeller
\( V'_{xi} \) = axial induced velocity in prop wash at some distance downstream
\( V_{xi} \) = axial induced velocity in plane of propeller
\( V'_{xi} \) = axial induced velocity in prop wash at some distance downstream
\( V_\infty \) = freestream velocity
\( V_{\infty,x} \) = component of freestream velocity parallel to rotation axis of propeller
\( w \) = tangential velocity
\( w_{ss} \) = self-similar tangential velocity profile of prop wash, Eq. (3.48)
\( \Delta w_{eq} \) = equivalent turbulent jet tangential velocity, Eq. (3.43-44)
\( x \) = axial distance downstream from propeller
\( x_e \) = length of zone of flow establishment
\( x^* \) = dimensionless downstream distance, Eq. (3.25)
\( \beta_G \) = Gaussian jet spread rate
\( \beta_s \quad = \quad \) square jet spread rate, Eq. (3.26)
\( \eta \quad = \quad \) percent distance through zone of flow establishment, Eq. (3.54)
\( \rho \quad = \quad \) freestream density

3.2 Background

In order to accurately model the effects of the propeller on lifting surfaces, an
accurate model of the propwash itself must be created. Multiple low-order methods for
modeling the prop wash have been proposed with varying degrees of accuracy\textsuperscript{1,2,7,28–31}. A
few of the more notable methods are briefly mentioned here.

Conway modeled the inviscid propeller slipstream as a combination of vortex
systems, which allowed for the analytical calculation of the inviscid slipstream
characteristics for simple blade loading case\textsuperscript{28}. Alba used this work done by Conway to
develop a simplified axial velocity scaling factor to show the acceleration of the inviscid
slipstream\textsuperscript{29}. Veldhuis used momentum theory to create a slipstream contraction ratio to
quickly and simply calculate the velocities in the inviscid slipstream\textsuperscript{1}. Khan used
correlations and observations from boat screws to make a semi-empirical model for a
turbulent prop wash\textsuperscript{7}. Stone applied conservation of mass and conservation of angular
momentum to a series of annular stream tubes to model the acceleration and contraction of
the inviscid slipstream\textsuperscript{5}. However, one flaw with existing methods is that they often ignore
viscosity or represent its effects inaccurately.

To more accurately model the development of the prop wash, the effects of
viscosity must be accounted for. While the effects of viscosity on prop wash development
are not very well understood, the characteristics and development of turbulent jets have
been extensively researched and are well understood. Although the driving physics are
dissimilar in several ways, observations of the effects of viscosity on a turbulent jet can provide valuable insight into how viscosity affects the slipstream of a propeller. Ultimately, observations on the behavior of turbulent jets are used to apply a number of turbulent corrections to an inviscid slipstream model to create a novel, low-order prop wash model that more closely matches experimental data.

### 3.2.1 Inviscid slipstream model using Stone’s method

The method presented in this section was originally developed by Stone\(^5\) and is reiterated here for clarity. Neglecting the effects of viscosity, the slipstream of a propeller can be closely modeled by applying conservation of mass and angular momentum to a series of annular stream tubes. This method works by determining the necessary dimensions of these stream tubes and the necessary velocities within them required to conserve mass and angular momentum. This results in a slipstream that accelerates and contracts as it develops downstream.

As a starting point for Stone’s method, the axial and tangential induced velocities must be known at a number of radial nodes in the plane of the propeller. These induced velocities can be found using various methods, such as the numerical blade element model previously described in Chapter 2. Once the induced velocities are known at the plane of the propeller, the axial induced velocity is scaled by the slipstream development factor, as proposed by McCormick\(^6\). This slipstream development factor reflects the increase in the axial induced velocity due to the propeller as the slipstream progresses downstream and is given as

\[
k_d = 1 + \frac{x}{\sqrt{x^2 + R^2}}
\]  

\text{(3.1)}
The slipstream development factor progresses from 1 to 2 as the distance downstream approaches infinity, thus reflecting the expected slipstream development based on classic propeller momentum theory. As the axial induced velocities in the slipstream are scaled by the slipstream development factor, the inviscid slipstream must also change its diameter so that mass flux is conserved.

To determine the amount of contraction of the slipstream, the slipstream is divided into a number of concentric, annular stream tubes whose initial inner and outer radii lie in the plane of the propeller on adjacent nodes at which the induced velocities were determined. The initial mass flow rate in a given annular stream tube is calculated from the radii of these nodes and the axial induced velocities in the plane of the propeller as

$$m_m = \rho A_m V_m = \rho \pi \left( r_{m+1}^2 - r_m^2 \right) \left( \frac{V_{ix_{m+1}} + V_{ix_m}}{2} + V_{w,x} \right)$$

Likewise, the mass flow rate through the same annular stream tube at some distance downstream is given as

$$m'_m = \rho \pi \left( r_{m+1}^2 - r_m^2 \right) k_d \left( \frac{V_{ix_{m+1}} + V_{ix_m}}{2} + V_{w,x} \right)$$

Note that the axial induced velocity has been multiplied by the slipstream development factor to reflect the acceleration of the slipstream. This increase of the velocity in the stream tube requires that the area of the stream tube be decreased to conserve mass flow. This is done by decreasing the radial position of the outer node, $r'_{m+1}$. To determine the amount of contraction, Eq. (3.2) and Eq. (3.3) are equated and rearranged as

$$r'_{m+1} = \sqrt{r_m^2 + K_d \left( r_{m+1}^2 - r_m^2 \right)}$$
\[ K_r = \left( \frac{2V_{x,i} + V_{x,i+1} + V_{x,i}}{2V_{x,i} + \beta V_{x,i+1} + V_{x,i}} \right) \]  

Therefore, to determine the radial positions of the annular stream tube nodes at some point downstream, the slipstream must be evaluated from the inside out. To do this, the radius of the innermost node is set to the radius of the nacelle. The radial position of the next node is then determined so that mass is conserved in the annular stream tube between these first two nodes. Once the radius of this second node is determined, the radius of the third node is calculated to satisfy conservation of mass in the stream tube between the second and third nodes. This same procedure is followed for the remaining nodes until the radius of the outermost node has been determined. This method is shown in Figure 3.1 and as

\[ r_{m+1} = r_{\text{nacelle}} \sqrt{\frac{m^2 + K_r (r_{m+1}^2 - r_m^2)}{r_m^2 + K_r (r_{m+1}^2 - r_m^2)}} \]

\[ m = 2 \ldots n \]  

**Fig. 3.1 Radial notation for Stone’s inviscid slipstream model.**
Once the radial positions of the streamtube nodes are determined, the velocity profile in the slipstream can be determined. As previously noted, the axial velocity in the slipstream is determined by multiplying the axial velocities in the plane of the propeller by the slipstream development factor. For the tangential velocity profile in the slipstream, it is first noted that the magnitude of the tangential velocity jumps to twice its at propeller plane value immediately behind the propeller. The tangential velocity is then scaled to ensure conservation of angular momentum within the annular stream tubes. Thus, the axial and tangential velocities in the slipstream can be expressed as:

\[
V'_{xi,n} = k_d V_{xi,n}
\]

\[
V'_{ti,n} = 2V'_{ti,n} \left( \frac{r_m}{r'_m} \right)
\]

Finally, the outer radius of the inviscid slipstream is defined as \( R' \). Thus, \( R' = r'_n \).

### 3.2.2 Turbulent Jets

As previously mentioned, the characteristics and development of turbulent jets have been extensively studied and modeled for decades. Observing the behavior of turbulent jets, parallels can be drawn to the development of the prop wash of a propeller. In this effort, a sound understanding of turbulent jets must first be obtained. Three cases are here considered; a jet into a stagnant ambient fluid, a jet into a coflowing fluid, and a jet with swirl. Each of these cases builds on the others and must be fully understood to develop a prop wash model using observations from turbulent jets. The analysis of the first two cases is drawn from Lee and Chu and the analysis of the last case is drawn predominantly from Rajaratnam.

It is important to note that all analysis in this section is time averaged. As the jet is
turbulent, there are strong eddies that make an instantaneous analysis of the jet very
difficult. However, when the properties of the jet are time-averaged, clear trends become
apparent that can be readily analyzed.

3.2.2.1 Turbulent jet in stagnant ambient fluid

In this case, a turbulent jet is issuing from a hole of diameter $D$ into a stagnant
ambient fluid with a uniform velocity $u_o$, as shown in Figure 3.2. As also seen in Figure
3.2, the development of the turbulent jet can be divided into two regions: the zone of flow
establishment (ZFE) and the zone of established flow (ZEF).

Fig. 3.2 Development of turbulent jet in stagnant ambient fluid.

In the zone of flow establishment, turbulence has a strong effect at the edges of the
jet profile where the velocity gradient from the jet to the stagnant ambient is very strong.
At these edges, a turbulent mixing layer forms where the momentum from the jet is
transferred to the stagnant ambient fluid. The profile of this mixing layer is assumed to closely match a Gaussian profile and the thickness of this mixing layer is assumed to increase linearly as

$$b = \beta_G x$$  \hspace{1cm} (3.9)$$
where $\beta_G$ is defined as the Gaussian jet spread rate and $b$ is the width at which the axial component of the velocity is equal to $1/e$ of the centerline value$^{13}$. Extensive measurements by Albertson et al. have found that this jet spread rate to equal approximately 0.114$^{13}$. However, as will be shown later, this jet spread rate can be altered by factors such as swirl.

Thus the velocity profile for a turbulent jet in a stagnant ambient fluid can be described by the following equations$^{13}$. In the zone of flow establishment,

$$u = u_o; r \leq R_{pc}$$  \hspace{1cm} (3.10)$$

and in the zone of established flow,

$$u = u_m \exp \left[-\left(\frac{r}{b}\right)^2\right]; r \geq R_{pc}$$  \hspace{1cm} (3.11)$$

and in the zone of established flow,

$$u = u_m \exp \left[-\left(\frac{r}{b}\right)^2\right]$$  \hspace{1cm} (3.12)$$

and in the zone of established flow,

$$u_m = u_o \frac{1}{\sqrt{2\beta_G}} \left(\frac{x}{D}\right)^{-1}$$  \hspace{1cm} (3.13)$$

The potential core radius, $R_{pc}$, is assumed to decrease linearly from $R_{pc}=D/2$ to $R_{pc}=0$ over the length of the zone of flow establishment. The length of the zone of flow establishment, $x_e$, is determined from Eq. (3.14). Inserting the experimentally found value of $\beta_G = 0.114$, the midline velocity, $u_m$, is found to equal the potential core velocity, $u_o$, at about $x = 6.2D$. This value agrees well with the experimentally observed potential core
length. This model for a turbulent jet in a stagnant ambient fluid has been extensively tested and been shown to match closely with experimental results.

Throughout this entire process, momentum flux must be conserved. This momentum flux is defined as

$$M = \int u^2 dA = 2\pi \int u^2 rdr$$  \hspace{1cm} (3.14)

It should also be noted that although momentum flux is conserved, the mass flux is not conserved because some of the ambient fluid surrounding the jet becomes entrained in the jet through the turbulent mixing.

### 3.2.2.2 Turbulent jet in coflowing fluid

Now that the simple case of a turbulent jet in a stagnant ambient fluid has been considered, the more complicated case of a turbulent jet in a coflowing fluid may be considered. In this case, the turbulent jet, with uniform velocity $u_o$, is inserted into an ambient flow of velocity $V\infty$, moving in the same direction as the jet. Note that $u_o$ is the total velocity of the jet relative to a stationary point, not relative to the coflowing fluid around it. This flow and the associated notation is shown in Figure 3.3.

This case has several notable differences from the jet in a stagnant fluid. First, for an incompressible turbulent jet in coflow, the specific excess momentum flux of the jet, not simply the specific momentum flux, is conserved\(^{14}\). This specific excess momentum flux is defined as

$$M_e = \int u(u - V\infty) dA$$  \hspace{1cm} (3.15)
In Eq. 3.15, \( u \) is the total velocity of the fluid as a function of radius. Alternately, using the notation shown in Figure 3.3, this specific excess momentum flux can be expressed as

\[
M_e = 2\pi \int_0^{D/2} \Delta u (\Delta u + V_{\infty,x}) r dr
\]  
(3.16)

Based on this specific excess momentum flux and the coflow velocity, a characteristic length scale, \( l_m^* \), is defined as\(^{14}\)

\[
l_m^* = \sqrt{M_e / V_{\infty,x}}
\]  
(3.17)

The second notable difference is that, unlike the jet in a stagnant fluid, the width of the jet is not assumed to expand linearly in the zone of established flow. Using a Lagrangian approach implemented by Lee and Chu\(^ {14}\), the expansion of the jet in the zone of established flow as a function of downstream distance can be found using a pair of differential
equations. However, before these equations are derived, the concept of a top-hat profile must be introduced.

As previously stated, the most important defining characteristic of a turbulent jet is the excess momentum flux. This excess momentum flux must be conserved throughout the development of the jet and influences the expansion rate of the jet. As such, instead of attempting to perform calculations using the Gaussian profile as previously described, a turbulent jet can be effectively modeled using a top-hat profile of equivalent excess momentum to simplify the analysis\textsuperscript{13,14}. This top-hat profile has a uniform velocity across its surface described as

\[
\begin{aligned}
  u &= V_{\infty,t} + \begin{cases} 
    \Delta U; r \leq B \\
    0; r > B 
  \end{cases} 
\end{aligned}
\]  

(3.18)

where \( \Delta U \) is the excess velocity of the top-hat profile and \( B \) is the half-width as shown in Figure 3.3. By equivalence of mass and momentum fluxes, it can be shown that the characteristics of this top hat profile can be equated to the Gaussian profile through the following relations

\[
\Delta U = \frac{\Delta u_m}{2} 
\]  

(3.19)

\[
B = \sqrt{2}b 
\]  

(3.20)

Since this top-hat profile is truly equivalent to the actual Gaussian profile, it can be used to determine the spreading rate of the turbulent jet. Using the Lagrangian method developed by Lee and Chu\textsuperscript{14}, the following set of differential equations is developed

\[
U'^2 + U^* - \frac{1}{\pi B^2} = 0 
\]  

(3.21)
\[ \frac{dB^*}{dx^*} = \beta_S \frac{U^*}{1+U^*} \]  \hspace{1cm} (3.22)

where the following non-dimensional parameters are used

\[ U^* = \Delta U / V_{m,x} \]  \hspace{1cm} (3.23)

\[ B^* = B / l_m^* \]  \hspace{1cm} (3.24)

\[ x^* = x / l_m^* \]  \hspace{1cm} (3.25)

and where

\[ \beta_S = \sqrt{2} \beta_G \]  \hspace{1cm} (3.26)

Subjected to the correct initial conditions, these differential equations can be integrated to determine the width of the top-hat profile as the slipstream develops downstream. As previously stated, the differential equations presented above apply only to the zone of established flow, where the velocity has already reached a self-similar, Gaussian profile. Thus, the initial conditions for solving these differential equations must be evaluated at the beginning of the zone of established flow.

To determine these initial conditions, the length of the zone of flow establishment must first be determined. The length of the zone of flow establishment, or the point downstream at which the zone of established flow begins, is determined to be

\[ x_e = D \frac{\sqrt{1 + V_{m,x} / u_o}}{\beta_S (1 - V_{m,x} / u_o)} \]  \hspace{1cm} (3.27)

It can be seen that this equation also holds for the case of a turbulent jet in a stagnant ambient fluid \((u_a = 0)\). In this case, \(x_e \approx 6.2D\), which is the same as the result obtained from Eq. (3.14). Next, the top-hat half width, \(B\), at the end of the zone of flow establishment is determined to be
\[
B|_{x=r} = \frac{D}{\sqrt{1 + V_{x,x}/u_o}} \tag{3.28}
\]

With these two initial values, the set of differential equations in Eqs. (3.21-3.22) can be integrated to determine the velocity profile of the jet in the zone of established flow.

For the profile in the zone of flow establishment, the mixing layer is assumed to expand linearly in a similar fashion to the stagnant case. However, unlike the stagnant case, the rate of expansion is determined by a linear interpolation between 0 and the top-hat half width given in Eq. (3.28)\(^{14}\). Thus, the radius of the potential core and the velocity profile in the zone of flow establishment are described by the following equations.

\[
R_{pc} = \frac{D}{2} \left(1 - \frac{x}{x_e}\right) \tag{3.29}
\]

\[
u = u_o; r \leq R_{pc} \tag{3.30}
\]

\[
u = V_{x,x} + \Delta u \exp\left[-\left(\frac{r - R_{pc}}{b}\right)^2\right]; r > R_{pc} \tag{3.31}
\]

### 3.2.2.3 Turbulent jet with swirl

The previously mentioned cases of a turbulent jet in a stagnant ambient fluid and a turbulent jet in a coflowing fluid are both well understood and modeled with relative ease. The third case, of a turbulent jet with swirl, is not as easily understood or modeled. When swirl is added to a turbulent jet, it can have multiple effects, the most notable of which include a faster spread rate and a more rapid decay of the velocity as it progresses downstream\(^{11}\). Additionally, the initial profile of the swirl as well as the relative strength of the swirl can strongly influence the jet’s development and its velocity profiles far downstream\(^{33}\). Because there are so many factors that influence the development of a
turbulent jet with swirl, they are not fully understood or modeled. However, several basic observations have been made that will be useful in the development of turbulent corrections for a propeller slipstream.

From Rajaratnam\textsuperscript{11}, it is shown that the axial momentum flux plus the pressure must be conserved in the axial direction in a turbulent jet with swirl

\begin{equation}
\frac{d}{dx} \int_{0}^{\infty} (p + \rho u^2) r dr = 0
\end{equation}

The pressure term in Eq. (3.32) can be related to the tangential velocity, \( w \), of the swirling jet such that

\begin{equation}
\frac{d}{dx} \int_{0}^{\infty} \rho \left( u^2 - \frac{w^2}{2} \right) r dr = 0
\end{equation}

Therefore, the value of the integral in Eq. (3.33) must be conserved in the axial direction.

As can be seen by comparing this expression with Eq. (3.14), this expression holds true for a purely axial jet where \( w(r)=0 \). This expression can also be expanded to reflect the conservation of only the excess momentum in a coflowing jet. Thus, this expanded equation describing the specific axial momentum flux of a swirling turbulent jet is given as

\begin{equation}
M = 2\pi \int_{0}^{\infty} \left( u (u - V_{x,\infty}) - \frac{w^2}{2} \right) r dr
\end{equation}

The analysis by Rajaratnam\textsuperscript{11} also shows that the specific angular momentum flux, \( L \), is also conserved in the axial direction. This angular momentum flux is defined as

\begin{equation}
L = 2\pi \int_{0}^{\infty} (uw) r^2 dr
\end{equation}

Using Eq. (3.34) and Eq. (3.35), together with the initial radius of the jet, \( R_0 \), a dimensionless parameter known as the swirl number\textsuperscript{11}, \( S \), is formed as
Experimental investigations by multiple individuals have shown that this swirl number is an important parameter that can be used to characterize a swirling turbulent jet. Experimental studies have shown that the jet angle, \( \alpha^o \), varies almost linearly with variation in the swirl number\(^{11} \). This jet angle can in turn be related back to the jet spread rate to give the jet spread rate as a function of the swirl number as

\[
\beta_G = \frac{\tan \left( \frac{\pi}{180} \left( 4.8 + 14S \right) \right)}{\sqrt{-\ln(0.5)}}
\]  

This jet spread rate can be separated into the axial and tangential contributions by identifying the spread rate present when there is no swirl and then subtracting that value from the amount determined by Eq. (3.37). Thus, the axial and tangential spread rates are given as

\[
\beta_{Gx} = \frac{\tan \left( \frac{4.8\pi}{180} \right)}{\sqrt{-\ln(0.5)}}
\]  

\[
\beta_{Gt} = \frac{\tan \left( \frac{\pi}{180} \left( 4.8 + 14S \right) \right)}{\sqrt{-\ln(0.5)}} - \beta_{Gx}
\]  

These jet spread rates can then be used in conjunction with the jet spreading hypothesis presented by Chu and Lee\(^{12} \). This jet spreading hypothesis assumes that the change in the width of the shear layer, in a Lagrangian frame of reference, is proportional to the relative velocity of the jet element and its surroundings. This is expressed as
\[ \frac{D\tilde{b}}{Dt} = |\tilde{u}| \frac{d\tilde{b}}{dx} = \beta_x |\Delta\tilde{u}_x| + \beta_t |\Delta\tilde{u}_t| \tag{3.40} \]

where \( \tilde{u} \) is the characteristic total velocity and \( \Delta\tilde{u}_x \) and \( \Delta\tilde{u}_t \) are the characteristic axial and tangential excess velocities respectively. This equation can ultimately be rearranged to determine the rate of change of the width of the mixing region. Note that as the jet progresses downstream, its velocities will decay as the jet continues to spread, thus resulting in a change in the jet spread rate. Thus, Eq. (3.40) must be used in conjunction with the equations for conservation of axial and angular momentum flux to create a system of equations. This system of equations must be integrated as a differential equation to determine the development of the jet as it progresses downstream.

While this section does not provide a complete analysis of swirling turbulent jets, the key observations presented above are helpful in the development of the necessary corrections to create a turbulent prop wash model.

### 3.3 Key characteristics of inviscid slipstream model

As previously stated, the observations made on the behavior and development of turbulent jets are used to create a number of turbulent corrections that are applied to an inviscid propeller slipstream model. Stone’s method, as previously presented, is used to model this inviscid propeller slipstream\(^5\). In order to apply the turbulent corrections, several observations must first be made regarding the inviscid slipstream.

First, it was previously stated that the excess axial momentum flux is conserved in a turbulent jet as given in Eq. (3.15). This is true for a turbulent jet, where there is little to no variation in the pressure in the axial direction. However, this is not true for a propeller slipstream. In a propeller slipstream, there is a step increase in the pressure at the plane of
the propeller that then decreases to the ambient pressure downstream\(^{20}\), as shown in Figure 3.4. Hence, it is more accurate to say that the axial excess momentum plus the pressure must be conserved as previously shown in Eq. (3.32).

![Diagram of pressure increase across plane of propeller](image)

**Fig. 3.4 Pressure increase across plane of propeller.**

The combination of this pressure gradient and the helical vortex system created by the propeller causes the acceleration and contraction of the propeller slipstream as it progresses downstream. In Stone’s method\(^5\), this is reflected using the slipstream development factor\(^6\) to scale the induced velocities downstream.

As shown, the axial momentum alone is not conserved as the slipstream progresses downstream due to the driving vortex system and pressure gradient. However, it is assumed that at any point downstream, the axial momentum flux of the inviscid slipstream must equal the axial momentum flux of the turbulent prop wash at the same point after all the turbulent corrections have been applied. Therefore, before applying any of the turbulent corrections, the axial excess momentum flux of the inviscid slipstream at the desired distance downstream must be calculated as

\[
M' = 2\pi \int_0^R \left[ V'_{xi} \left( V'_{xi} + V_{\infty,x} \right) - \frac{V'_a^2}{2} \right] rdr
\]

(3.41)
Additionally, the tangential momentum flux of the inviscid slipstream must be calculated and conserved between the inviscid slipstream and turbulent prop wash. The angular momentum flux of the inviscid slipstream is calculated according to Eq. (3.42).

\[
L' = 2\pi \int_0^\infty \left( (V'_{x_i} + V'_{w,x}) r' \right) r'^2 dr' \tag{3.42}
\]

Next, it is useful to calculate the velocity magnitudes of an equivalent swirling turbulent jet. This equivalent swirling jet has the same radius as the inviscid slipstream and has uniform axial and tangential velocities that give it the same axial and angular momentums as the inviscid slipstream. The axial and tangential velocity magnitudes of this equivalent jet are calculated by solving the system of equations given in Eq. (3.43) and (3.44) for \( \Delta u_{eq} \) and \( \Delta w_{eq} \).

\[
\pi \left[ \Delta u_{eq} \left( \Delta u_{eq} + V_{w,x} \right) - \frac{\Delta w_{eq}}{2} \right] R'^2 = M' \tag{3.43}
\]

\[
2\pi \left[ (\Delta u_{eq} + V_{w,x}) \Delta w_{eq} \right] R'^3 = L' \tag{3.44}
\]

These velocities are used as characteristic velocities for the propeller slipstream in the zone of flow establishment.

3.4 Incorporating the effects of turbulence

As previously mentioned, the effects of viscosity and turbulent mixing on the slipstream of a propeller are closely modeled by applying a number of turbulent corrections to the inviscid slipstream model based on observations of turbulent jets. Similar to a turbulent jet, the turbulent prop wash can be divided into two regions, the zone of flow establishment and the zone of established flow. In the zone of flow establishment, turbulent mixing acts around the edges of the slipstream as well as through the core of the slipstream.
to change the velocity profiles from their at-propeller values to self-similar profiles at the end of the zone of flow establishment. In the zone of established flow, these self-similar profiles are propagated downstream as the slipstream expands and decays. Therefore, the first step in the process of modeling the turbulent prop wash must be to determine the length of the zone of flow establishment, $x_e$.

### 3.4.1 Length of zone of flow establishment

Similar to a turbulent jet, a turbulent mixing region forms at the outer edges of a propeller slipstream due to the large velocity gradient between the slipstream and the ambient air. The end of the zone of flow establishment is defined as the point at which this mixing region has expanded sufficiently to penetrate to the centerline of the slipstream. At this point, the time averaged velocity profiles have become fully self-similar and the zone of established flow begins. In order to calculate the length of the zone of flow establishment, several expressions are developed describing the width of the mixing region at the end of the zone of flow establishment. Using these several expressions, a numerical root finding method can be used to find the value of $x_e$ that satisfies both expressions.

The first expression comes from the spreading hypothesis as previously presented in Eq. (3.40). To apply this spreading hypothesis to the propeller slipstream, the equivalent jet velocities previously calculated in Eq. (3.43) and Eq. (3.44) are used as the characteristic velocities of the slipstream as a function of the distance downstream. Thus, the average components of the excess velocity in the mixing region are $\Delta v_x/2$ and $\Delta v_y/2$. Inserting these values into the spreading hypothesis gives the following equation for the rate of expansion of the mixing region,
\[
\frac{db}{dx} = \frac{\beta_x \left| \frac{\Delta u_{eq}}{2} \right| + \beta_t \left| \frac{\Delta w_{eq}}{2} \right|}{\sqrt{\left( V_{w,x} + \frac{\Delta u_{eq}}{2} \right)^2 + \left( \frac{\Delta w_{eq}}{2} \right)^2}}
\]  

(3.45)

This expression is then integrated over the length of zone of flow establishment to determine the width of the mixing region at the end of the zone of flow establishment as

\[
b(x_e) = \int_0^{x_e} \left[ \frac{\beta_x \left| \frac{\Delta u_{eq}}{2} \right| + \beta_t \left| \frac{\Delta w_{eq}}{2} \right|}{\sqrt{\left( V_{w,x} + \frac{\Delta u_{eq}}{2} \right)^2 + \left( \frac{\Delta w_{eq}}{2} \right)^2}} \right] \, dx
\]

(3.46)

Note that all the variables in this equation except the freestream velocity, \( V_{w,x} \), are functions of the distance downstream, \( x \), and must therefore be included in the integration.

The second expression uses conservation of axial and angular momentums at the end of the zone of flow establishment to determine the width of the mixing region. To do this, equations must first be set forth describing the self-similar axial and tangential velocity profiles. Based on the observation of turbulent jets, it is assumed that the axial velocity will reach a self-similar profile that can be closely described using a Gaussian curve. Thus, the axial velocity profile at the transition plane and through the zone of established flow is given as

\[
\bar{u}_{ss} = \bar{V}_{w,x} + \Delta u_m \exp \left[ -\left( \frac{r}{b} \right)^2 \right]
\]

(3.47)

where \( \Delta u_m \) is the centerline excess velocity and \( b \) is the Gaussian width of the profile.

Because turbulent jets with swirl are not yet well understood or modeled, the self-similar profile for the tangential velocity comes from observation of measured propeller
slipstream data. Although the data observed does not show the development of the slipstream into the zone of established flow, it can be seen that the velocity profile is approaching a shape that can be generally described using two straight line segments. Thus, the tangential velocity profile at the transition plane and through the zone of established flow is given as

\[
W_{sx} = \begin{cases} 
W_m \frac{r}{r_{wm}}; & r \leq r_{wm} \\
W_m \frac{(r-c)}{(r_{wm} - c)}; & r_{wm} < r < c \\
0; & r \geq c 
\end{cases}
\] (3.48)

\[c = F_{w} b\] (3.49)

\[r_{wm} = F_{wm} c\] (3.50)

where the factors \(F_w\) and \(F_{wm}\) are chosen to fit experimental data, as discussed later in Section 3.5. The shape of these self-similar profiles are shown in Figure 3.5.

![Fig. 3.5 Self-similar axial and tangential velocity profiles.](image)

Using these self-similar profiles, a system of equations based on the momentum equations can be developed and solved to determine the width of the mixing region at the
end of the zone of flow establishment. This system of equations is created by inserting the self-similar axial and tangential velocity profiles given in Eq. (3.47) and Eq. (3.48) into the expressions for the axial and angular momentum given in Eq. (3.34) and Eq (3.35). The momentums of these profiles are then compared to the momentums of the inviscid slipstream at $x_e$ as

$$2\pi \int_0^\infty \left[ u_{ss}(u_{ss} - V_{w,ss}) - \frac{W_{ss}^2}{2} \right] r dr - M'(x_e) = 0$$

(3.51)

$$2\pi \int_0^\infty [u_{ss} W_{ss}] r^2 dr - L'(x_e) = 0$$

(3.52)

In Eq. (3.51) and Eq. (3.52), there are currently three unknowns that come from the definitions of the self-similar velocity profiles: $u_m$, $b$, and $w_m$. As there are only two equations, an assumption must be made about the value of one of these unknowns in order to solve this system of equations. The assumption that is made is that the centerline axial velocity, $u_m$, is related to the axial equivalent jet velocity, $\Delta u_{eq}$, through the following equation.

$$u_m(x_e) = F_{um} \Delta u_{eq}$$

(3.53)

where $F_{um}$ is chosen to fit experimental data, as discussed later in Section V. With this assumption made, the system of equations in Eq. (3.51) and Eq. (3.52) can be solved for $b$ and $w_m$ at the proposed $x_e$.

There are now two different methods for determining the width of the mixing region at the end of the zone of flow establishment; one using the spreading hypothesis and another using conservation of momentum. Both of these methods are functions of the length of the zone of flow establishment, $x_e$. Thus, in order to determine the value of $x_e$, a numerical root finding method must be employed to determine the value of $x_e$ that produces
the same mixing region width using both methods. This value of $x_e$ is considered to be the length of the zone of flow establishment.

### 3.4.2 Velocity profiles in zone of flow establishment

Observation of experimental measurements of the prop wash velocities by Deters\(^\text{10}\) shows several important trends that must be reflected in a turbulent prop wash model. Figure 3.6 shows the axial and tangential velocity profiles measured in the slipstream produced by a GWS 5x4.3 propeller\(^\text{10}\). This data shows the development of the slipstream through much of the zone of flow establishment and is exemplary of trends from other propellers.

![Fig. 3.6 Axial and tangential velocity distribution in slipstream of GWS 5x4.3 propeller.](image)

The first important trend to note is that as the slipstream progresses downstream, the axial velocity distribution as shown in Figure 3.6 progresses from a unique shape in the plane of the propeller towards a self-similar shape, similar to the development of a turbulent jet. Second, during its development, the effects of turbulence are visible in two regions; close to the axis and on the outside edge of the slipstream. Near the axis, turbulence causes the momentum within the slipstream to diffuse and draw up the axial velocity deficit at the
axis. At the outside edges of the prop wash, a mixing region forms, similar to that of a turbulent jet, and causes the prop wash to expand as it entrains ambient fluid. Third, the point of max velocity steadily progresses from the outer portion of the slipstream towards the axis. This also shows the tendency of the slipstream towards a fully self-similar axial velocity profile. Similar trends are visible in the tangential velocity profiles.

In order to model these trends, multiple corrections are applied to the axial and tangential velocity profiles of the inviscid slipstream. The corrections applied to the axial velocity profile are considered first. These corrections are described below using a number of temporary velocity profiles, $u_{\text{temp}}$, to show the intermediate steps used to arrive at the final velocity profile. Through all of these steps, the percent distance through the zone of flow establishment is used repeatedly. This percent distance is given as

$$\eta = \frac{x}{x_e} \quad (3.54)$$

The first step is to blend the original axial velocity profile with the velocity magnitude of the equivalent jet. This reflects the effects of turbulence as it draws up the velocity deficit at the centerline and begins to smooth out any sharp velocity gradients in the interior of the prop wash. This first step is given as

$$u_{\text{temp},1} = (1 - \eta)\left[V'_{x_1} + V_{x_e,1}\right] + \eta \Delta u_{eq} \quad (3.55)$$

Next, the radius of the pseudo-potential core is determined. Because there are velocity gradients within the core of the slipstream, it is not truly a potential core as in a turbulent jet. Instead, it is only a reflection of how far into the slipstream the outer mixing region has penetrated. Additionally, it should be noted that because the inviscid slipstream contracts as it progresses downstream, the pseudo-potential core also contracts with the slipstream. Therefore, the radius of the pseudo-potential core is considered to be a
percentage of the outer radius of the inviscid slipstream. This radius is given as

\[ R'_{pc} = (1 - \eta)R' \]  \hspace{1cm} (3.56)

Once the radius of the potential core is determined, the mixing region at the outside edge of the slipstream is modeled. As this mixing region is modeled by a Gaussian curve, the max velocity and the width of this mixing region must be determined. The max velocity is found by interpolating along the first temporary axial velocity profile, \( u_{temp,1} \), to determine the velocity at the edge of the pseudo-potential core as

\[ u_{pc} = u_{temp,1}(R'_{pc}) \]  \hspace{1cm} (3.57)

The width of the mixing region is assumed to expand linearly through the zone of flow establishment. This is not entirely true, as spreading rate is affected by the magnitude of the characteristic velocities as shown in the spreading hypothesis. However, it is a reasonable approximation for the purposes of this model. Thus, the width of the mixing region is found by multiplying the blending factor, \( \eta \), by the mixing region half-width at the end of the zone of flow establishment, as previously calculated in Eq. (3.46). Thus, the mixing region width in the zone of flow establishment is given as

\[ b = \eta b(x_e) \]  \hspace{1cm} (3.58)

With these two values, the next temporary axial velocity profile is divided into regions inside and outside of the pseudo-potential core and defined as

\[ u_{temp,2} = u_{temp,1}, r \leq R'_{pc} \]

\[ u_{temp,2} = \left( u_{pc} - V_{\infty},x \right) \exp \left[ -\left( \frac{r - R'_{pc}}{b} \right)^2 \right] + V_{\infty},x, r \geq R'_{pc} \] \hspace{1cm} (3.60)

The final step in establishing the axial velocity profile is to scale it so that it has the same momentum as the inviscid slipstream. However, before finalizing the axial velocity
profile, the tangential velocity profile must also be considered. The tangential velocity profile will be developed in a similar fashion; using a number of temporary velocity profiles. The first temporary velocity profile is created by blending the inviscid velocity profile with the equivalent jet velocity as shown in Eq. (3.61).

$$w_{\text{temp},1} = (1 - \eta)w_n + \eta \Delta w_{eq}$$

(3.61)

Next, the radius of the mixing regions is determined. In order to model a smooth transition from the initial inviscid profile to the self-similar profile, two mixing regions are modeled; one that grows from the centerline and one that grows from the outer edge of the slipstream, as seen in Figure 3.7. These two mixing regions are the two regions where the velocity gradients are the strongest, thus resulting in turbulent mixing.

**Fig. 3.7 Mixing regions for development of tangential velocity profile.**

The radii of these mixing regions as seen in Figure 3.7 is established using the
percent distance through the zone of flow establishment using the following equations.

\[ r_1 = r_{wm}(x_e)\eta \tag{3.62} \]
\[ r_2 = (1 - \eta)R' + r_{wm}(x_e)\eta \tag{3.63} \]
\[ r_3 = (1 - \eta)R' + c(x_e)\eta \tag{3.64} \]

The second temporary profile can then be described using the following equations.

\[
\begin{align*}
\omega_{\text{temp},2} &= \begin{cases}
  \frac{r}{r_1} \omega_{\text{temp},1} & 0 \leq r \leq r_1 \\
  \frac{r - r_1}{r_2 - r_1} \frac{r_1}{r_2} \omega_{\text{temp},1} & r_1 < r < r_2 \\
  \frac{r - r_1}{r_3 - r_1} & r_2 \leq r < r_3 \\
  0 & c \leq r
\end{cases} \tag{3.65}
\end{align*}
\]

where

\[ \omega_{r1} = \omega_{\text{temp},1}(r_1) \tag{3.66} \]
\[ \omega_{r2} = \omega_{\text{temp},1}(r_2) \tag{3.67} \]

Finally, \( u_{\text{temp},2} \) and \( w_{\text{temp},2} \) must be scaled so that its axial and angular momentum fluxes match the axial and angular momentum fluxes of the inviscid slipstream at the same distance downstream. Two scaling factors, \( S_{fx} \) and \( S_{ft} \), are applied to the temporary velocity profiles as follows

\[
\begin{align*}
  u_{\text{scaled}} &= (u_{\text{temp},2} - V_{\omega,x})S_{fx} + V_{\omega,x} \tag{3.68} \\
  w_{\text{scaled}} &= S_{ft}w_{\text{temp},2} \tag{3.69}
\end{align*}
\]

The two scaling factors, \( S_{fx} \) and \( S_{ft} \), are chosen such that the axial and angular momen- tumoms of the scaled profiles, as calculated using Eq (3.34) and Eq (3.35), match the axial and angular momentums of the inviscid slipstream at the same point downstream, as
calculated using Eq. (3.41) and Eq. (3.42). This may be done using any of a number of two-dimensional root finding methods. Once the correct scaling factor is found, the velocity profiles at a given downstream distance in the zone of flow establishment are given in Eq. (3.68) and Eq. (3.69).

3.4.3 Velocity profiles in zone of established flow

Modeling the effects of turbulence in the zone of established flow is much simpler than modeling the effects in the zone of flow establishment. In the zone of established flow, the axial and tangential velocity profiles have already reached self-similar shapes. As the flow progresses downstream, these self-similar profiles simply continue to expand and their max velocity decays appropriately so that the axial and angular momentum flux are conserved.

The width and expansion rate of the prop wash in the zone of established flow are calculated using the spreading hypothesis given in Eq. (3.40). The spreading hypothesis allows the expansion rate of the prop wash to be solved based on the magnitudes of the characteristic velocities. However, as the prop wash develops downstream, the magnitudes of these characteristic velocities will decrease to conserve momentum as the prop wash expands. Because of these decreasing velocities, a system of equations must be developed and numerically integrated to calculate the development of the prop wash as it progresses downstream.

The first equation in this system of equations is the spreading hypothesis. For the spreading hypothesis in the zone of flow establishment, top hat profiles similar to those previously discussed in the sections on turbulent jets are used to find the characteristic velocities. These top hat profiles are described as
Using these top-hat velocities as the characteristic velocities, the spreading hypothesis takes the form

\[
\frac{dB}{dx} = \frac{\beta_1 |\Delta U| + \beta_2 |\Delta W|}{\sqrt{\left(V_{w,x} + \Delta U\right)^2 + (\Delta W)^2}}
\]

This spreading hypothesis forms the foundation of the system of equations that must be integrated in the ZEF. At each step of the numerical integration, the expansion rate of the prop wash is calculated. The prop wash width at a short distance downstream is then calculated based on this expansion rate. Then based on this new width, the top hat velocity magnitudes are reevaluated so that they conserve axial and angular momentum. Thus, as the spreading hypothesis is integrated downstream, the width increases at a non-constant rate and the characteristic top hat velocity magnitudes are calculated to conserve axial and angular momentum.

The initial conditions for this numerical integration are determined by calculating the characteristic top hat profiles at the beginning of the zone of established flow, \(x_e\). At this point, the top hat width, \(B\), and velocities, \(\Delta U\) and \(\Delta W\), are chosen so that they match the axial and angular momentum as well as the axial mass flux of the self-similar profiles. This is done by solving the following system of equations for \(B_{xe}, \Delta U_{xe}\), and \(\Delta W_{xe}\)

\[
2\pi \int_0^{B_{xe}} \left( \Delta U_{xe} \left( \Delta U_{xe} - V_{w,x} \right) - \frac{\Delta W_{xe}^2}{2} \right) r dr = M'(x_e)
\]

(3.73)
Using these initial conditions in conjunction with the spreading hypothesis, the top hat profiles characterizing the prop wash can be calculated from the end of the zone of flow establishment, $x_e$, to any distance downstream. Once these top hat profiles are determined, conservation of axial and angular momentums as well as conservation of mass flux can be applied to determine $b$, $\Delta u_m$, and $\Delta w_m$ of the corresponding self-similar velocity profiles. These self-similar profiles, expressed in Eq. (3.47) and Eq. (3.48), describe the axial and tangential velocity in the zone of established flow.

3.5 Semi-empirical corrections

The method presented above for modeling turbulent prop wash development provides good qualitative agreement with experimental measurements. However, in order to attain better quantitative agreement, a number of semi-empirical correction factors must be applied to this model. These correction factors apply to the momentum and development of the inviscid slipstream, the tangential velocity’s self-similar profile, the turbulent spread rates, and the centerline velocity at the end of the zone of flow establishment.

Most inviscid propeller slipstream models, including Stone’s method\(^5\), model the acceleration and contraction of the slipstream due to the trailing helical vortex system created by the propeller. This vortex system increases the velocity induced by the propeller from its initial value, in the plane of the propeller, to twice its initial value at some point infinitely far downstream\(^20\). This increase in the velocity of the inviscid slipstream, together with its corresponding contraction, causes a large increase in the axial momentum of the
slipstream as it progresses downstream.

However, observation of experimental data collected by the University of Illinois\textsuperscript{10} shows that this is not entirely true for an actual propeller. Observing the data collected, there is a clear initial increase in the momentum of the prop wash as it accelerates and contracts. However, this increase quickly stagnates and the momentum of the prop wash further downstream is often much lower than the momentum predicted by the inviscid slipstream model. This is likely due to the fact that when turbulence is considered, the trailing helical vortex system quickly breaks down as it progresses downstream. This in turn results in a lower momentum than that predicted by the inviscid slipstream model that assumes a vortex system that extends infinitely far downstream.

In order to account for this momentum difference between the inviscid slipstream model and experimental results, two simple modifications are made to McCormick’s slipstream development factor\textsuperscript{6} as given in Eq. (3.1). First, it is observed from experimental data that the actual slipstream initially accelerates and contracts faster than that predicted by the inviscid slipstream model. To account for this, a correction factor, $F_{kd}$, is added to the slipstream development factor as

$$
k_d = 1 + \frac{F_{kd} \cdot x}{\sqrt{x^2 + R^2}}
$$

(3.76)

This is done to accelerate the initial development. A value of $F_{kd} = 2$ was found to provide good agreement over the range of experimental data observed.

Second, after the initial contraction and expansion, the expected momentum increase in the experimental prop wash measurements quickly stagnates. As previously mentioned, this is likely due to a breakdown of the helical vortex system. To model this behavior, a development stagnation point, $x_{ds}$, is chosen at some short distance downstream
from the propeller. Up until this point, the slipstream development factor is calculated according to Eq. (3.76). However, after this point, where \( x > x_{ds} \), the slipstream development factor maintains the value it had at the development stagnation point. This causes the momentum and velocity profiles of the inviscid slipstream to become constant past this point. A value of \( x_{ds} = 0.1875D_p \) was found to provide good agreement over the range of experimental data observed.

As previously mentioned, because turbulent jets with swirl are not yet well understood or modeled, the shape of the self-similar tangential velocity profile of a swirling jet is unknown. Therefore, the self-similar profile described in Eq. (3.48) was chosen based on the observed development of the tangential velocity profiles in the experimental prop wash data. Additionally, the values of \( F_w \) and \( F_{wm} \) in Eq. (3.49) and Eq. (3.50) respectively are chosen to provide good agreement with experimental data. Values of \( F_w = 1.5 \) and \( F_{wm} = 0.1 \) were found to provide good agreement over the range of experimental data observed.

The spreading rates used in the turbulent prop wash model were pulled directly from observations on swirling turbulent jets. However, experiments have shown that the development of swirling turbulent jets is heavily dependent upon such things as initial tangential velocity distribution\(^{11,33}\). Further, it is expected that directly applying these spread rates to a turbulent prop wash model will result in discrepancies. Therefore, a correction factor is applied to both the axial and tangential spread rates to improve agreement with experimental data. With this correction factor, the new spread rates are given in Eq. (3.77) and Eq. (3.78). A value of \( F_\beta = 1.414 \) was found to provide good agreement over the range of experimental data observed.
\[ \beta_{Gx,prop} = F_\beta \beta_{Gx,jet} \]  
(3.77)

\[ \beta_{Gt,prop} = F_\beta \beta_{Gt,jet} \]  
(3.78)

The final correction factor that must be applied relates to the centerline velocity of the self-similar axial velocity profile at the end of the zone of flow establishment. As shown in Eq. (3.53), an assumption is made about the centerline velocity in order to solve the system of equations for the location of the end of the zone of flow establishment. A value of \( F_{um} = 0.8 \) was found to provide good agreement with the experimental data observed.

It should be noted once again that the correction factor values chosen were selected to improve general agreement with the observed experimental data, as presented in Section 3.6. Unfortunately, the range of experimental data available was rather limited. Therefore, further work could be done to improve these factors so that they have good agreement when compared to a wider range of experimental data.

### 3.6 Results of turbulent prop wash model

The results of the turbulent prop wash model presented above are here compared with experimental results obtained from the University of Illinois\(^\text{10}\). Deters, from the University of Illinois, tested a number of small-scale hobby propellers and obtained measurements of the prop wash velocities at a number of operating conditions\(^\text{10}\). Presented below are the results of the turbulent prop wash model compared with the experimental measurements obtained by Deters.

Figure 3.8 shows the axial and tangential velocity profiles predicted by the turbulent prop wash model for a GWS 5x4.3 propeller operating in a static thrust condition. As can be seen, the prop wash development qualitatively matches with the expected development, namely the transition toward self-similar profiles.
Fig. 3.8 Numerical model of development of prop wash of GWS 5x4.3 propeller.

When compared to experimental measurement, these results show good quantitative agreement over the range of data available. Figures 3.9–3.14 show the axial and tangential velocity profiles obtained from the turbulent prop wash model compared to experimental data for a number of propellers and operating conditions.

Fig. 3.9 Numerical prop wash vs. experimental measurements for GWS 5x4.3 propeller at $J=0$. 
Fig. 3.10 Numerical prop wash vs. experimental measurements for GWS 5x4.3 propeller at $J=0.52$.

Fig. 3.11 Numerical prop wash vs. experimental measurements for APC 4.2x2 propeller at $J=0$.

Fig. 3.12 Numerical prop wash vs. experimental measurements for DA4002 5x3.75 propeller at $J=0$. 
Fig. 3.13 Numerical prop wash vs. experimental measurements for DA4002 9x6.5 propeller at \( J=0 \).

Fig. 3.14 Numerical prop wash vs. experimental measurements for DA4002 9x6.5 propeller at \( J=0.64 \).

As can be seen, the level of agreement between the numerical and experimental results varies based on the propeller and the operating condition. This is likely due to a number of factors. One possible factor is that the initial velocity profile for the turbulent prop wash model is obtained using a blade element model. This can result in an initial velocity profile that is different from the actual initial velocity profile, thus changing the prop wash development. Additionally, the data obtained from Deters\textsuperscript{10} did not specify the uncertainty of the experimental measurements. Thus, it is possible that the results obtained from the numerical model could fall within the uncertainty of the experimental results.
It is also clear from the results presented above that the effects of turbulence are most clearly visible at low advance ratios. At high advance ratios, the freestream velocity carries the development of the prop wash far enough downstream that the effects of turbulence are not very apparent close behind the propeller. Thus, an inviscid model would be acceptable for analyzing the prop wash velocity at points reasonably close to a propeller with a high advance ratio. The true strength of the presented turbulent prop wash model becomes apparent when considering propellers operating at low advance ratios.

Figure 3.15 compares the results of the turbulent prop wash model to an inviscid slipstream model. As can be seen, when considering a propeller at a low advance ratio, the use of an inviscid slipstream model significantly over predicts the magnitude of the slipstream velocities and under predicts the radius of the slipstream. Using the inviscid slipstream model to find the influence of the prop wash on a lifting surface would result in a very concentrated and exaggerated alteration of the lift distribution. Therefore, the turbulent prop wash model is desirable because of its accuracy over a wider range of potential operating conditions.

**Fig. 3.15 Inviscid vs. turbulent prop wash model results for GWS 5x4.3 propeller at \( J=0 \).**
3.7 Grid resolution study of turbulent prop wash model

The convergence of the turbulent prop wash model was tested by varying the number of radial nodes and calculating the axial and angular momentums of the resulting prop wash velocity profiles at 1 diameter downstream. It should be noted that the number of radial nodes used in the turbulent prop wash model is the same as the number of nodes used in the blade element model and the propeller described in section 2.6 is also used here. Figure 3.16 shows the variation in the axial and angular momentums as a function of radial nodes.

![Graph showing variation in axial and angular momentums as a function of radial nodes.](image)

**Fig. 3.16 Axial and angular momentum as function of radial nodes.**

The percent error in these values was calculated using the value with the highest resolution as the reference point. These results can be seen in Figure 3.17. From this analysis, the convergence of the turbulent prop wash model is determined to have an order
ranging from 1.5–2.0. This order of convergence varies more significantly than the numerical blade element model’s convergence depending on the propeller geometry and operating condition. Based on this analysis, it is recommended that the same amount of radial nodes be used as was previously determined in the numerical blade element model grid resolution study. While this does result in a higher level of error, the turbulent prop wash model is inherently an approximation of the real turbulent prop wash behavior, so a small amount of error due to a lower resolution is an acceptable loss to maintain good computational efficiency.

Fig. 3.17 Percent error of momentums as function of radial nodes.
CHAPTER 4
NUMERICAL LIFTING LINE MODEL

4.1 Nomenclature

\( A_i \) = area of wing section \( i \)

\( a_k \) = unit vector in direction of axis of propeller \( k \)

\( \text{ARF} \) = axial reduction factor

\( \text{AR} \) = aspect ratio

\( C_{Li} \) = lift coefficient of wing section \( i \)

\( d \ell_i \) = directed differential vortex length of wing section \( i \)

\( dF_i \) = section aerodynamic force vector for wing section \( i \)

\( r_{i,j} \) = vector from 1\textsuperscript{st} node on section \( i \) to control point on section \( j \)

\( r_{i,j} \) = vector from 2\textsuperscript{nd} node on section \( i \) to control point on section \( j \)

\( r_{i,j} \) = magnitude of \( r_{i,j} \)

\( r_{i,j} \) = magnitude of \( r_{i,j} \)

\( \text{SRF} \) = swirl reduction factor

\( t_k \) = unit vector in direction of tangential velocity of propeller \( k \)

\( \mathbf{u}_\infty \) = unit vector in direction of freestream

\( \mathbf{v}_{ji} \) = velocity induced on section \( i \) by horseshoe vortex \( j \), Eq. (4.6)

\( V_{\text{rel},i} \) = local upstream velocity at section \( i \)

\( V_{\text{tot},i} \) = total velocity at section \( i \)

\( V_{\text{tot},i} \) = magnitude of total velocity at section \( i \)

\( V_t \) = tangential velocity in prop wash
\[ V_s = \text{axial velocity in prop wash} \]
\[ \alpha = \text{angle of attack} \]
\[ \delta = \text{flap deflection} \]
\[ \delta_{ij} = \text{Kronecker delta (1 if } i = j, 0 \text{ if } i \neq j) \]
\[ \rho = \text{freestream density} \]
\[ \Gamma_i = \text{circulation of wing section } i \]

4.2 Assumptions regarding propeller-wing interaction

Before considering the method for modeling the effects of the prop wash on lifting surfaces, it is important to understand a number of assumptions employed in the development of this model. As the goal of this research is the development of a computationally efficient, low-order model, several simplifying assumptions are made regarding the propeller-wing interaction in order to increase computational efficiency while maintaining a good level of accuracy.

First, it is assumed that the velocity field in the slipstream of the propeller is axisymmetric and time-averaged. This is not entirely physically accurate, but is a safe assumption for the purposes of this low-order model.

Second, it is assumed that the slipstream of the propeller progresses back in a straight line along the axis of the propeller. In reality, if there is an angle of attack between the free stream direction and the propeller axis, the free-stream velocity would deflect the propeller slipstream until the slipstream flowed downstream in the direction of the free-stream. However, since the numerical lifting line code used to analyze the aerodynamics of the wings is only valid at angles of attack below stall, the angle between the propeller axis and the free-stream direction will typically be small. Additionally, the propellers are
typically placed close to the wings, minimizing the distance over which the free-stream could deflect the propeller slipstream. Therefore, as long as these conditions are understood and followed, it can be safely assumed that the propeller slipstream progresses back in a straight line along the axis of the propeller.

Third, it is assumed that the interaction between the propeller and the wings is one-way; the propeller affects the wings, but the wings do not influence the propeller. This means that only the effects of the prop wash on the wing are modeled and that any downwash or upwash produced by the wings does not influence the propeller performance or the prop wash development. This assumption is reasonably safe because the angle of attack on the propeller mainly affects off-axis forces and moments produced by the propeller, but not the prop wash development. However, this assumption is mainly employed to reduce the computation time of this method. Were the propeller-wing interaction modeled as being two-way, it would require multiple iterations of both the propeller and lifting line models and would increase the computational cost significantly.

It is anticipated that employing these assumptions will decrease the level of accuracy of the proposed model. However, as this model is already of a low-order nature, the small decrease in accuracy caused by these assumptions is an acceptable loss in order to maintain the high computational efficiency of this method. If a higher level of accuracy is desired, other methods such as computation fluid dynamics or wind tunnel testing would be more desirable.

4.3 Numerical Lifting Line Method

A modern numerical lifting line method proposed by Phillips is used to model the aerodynamics of lifting surfaces\(^4,16\). This modern lifting line method is based on Prandtl’s
original Lifting-Line Theory\textsuperscript{34}, but includes modifications to make it applicable to any number of wings with arbitrary position and orientation. This method has been shown to accurately predict inviscid forces and moments of lifting surfaces at a fraction of the computation cost of other methods such as panel methods or computational fluid dynamics. Given below is a dimensional derivation of Phillips’ numerical lifting line method as presented by Hunsaker and Snyder\textsuperscript{3,15}.

In Phillips’ numerical lifting line method, a finite wing is modeled as a series of horseshoe vortices with one edge lying along the quarter chord of the wing and the trailing portions aligned with the freestream velocity, as seen in Figure 4.1. By relating the strength of each horseshoe vortex to the lift produced by a similar 2D airfoil section with the same local angle of attack, a series of equations can be created and solved to determine the forces and moments along the span of the wing.

![Fig. 4.1 Horseshoe vortices used in numerical lifting line method.](image-url)
Employing a 3D vortex lifting law, the differential force vector produced by a finite wing section, $i$, is

$$\mathbf{dF}_i = \rho \Gamma_i \mathbf{V}_i \times \mathbf{d} \ell_i$$

(4.1)

Additionally, the lift coefficient of a 2D airfoil section can be expressed as an arbitrary function of local angle of attack and flap deflection

$$C_{Li} = C_{Li}(\alpha_i, \delta_i)$$

(4.2)

From this lift coefficient, the magnitude of the differential force produced by a finite wing section is

$$dF_i = \frac{1}{2} \rho V_{tot}^2 A_i C_{Li}(\alpha_i, \delta_i)$$

(4.3)

As previously stated, the magnitude of the force produced by the 3D vortex lifting law in Eq. (4.1) is equated to the force produced by the 2D airfoil section in Eq. (4.3). Equating these two values and rearranging, the following expression is derived

$$2 \Gamma_i \left( \mathbf{v}_{rel} + \sum_{j=1}^{N} \Gamma_j \mathbf{V}_{ji} \right) \times \mathbf{d} \ell_i - V_{tot}^2 A_i C_{Li}(\alpha_i, \delta_i) = 0$$

(4.4)

Note that the velocity at section $i$ for the 3D vortex lifting law has been split into the local upstream velocity and the sum of the velocities induced by all of the horseshoe vortices in the system. The local upstream velocity differs from the freestream velocity in that it could incorporate velocity induced by prop wash or rotation of the wing about the center of gravity. For the 2D airfoil section force calculations, the total velocity magnitude is given as

$$V_{tot} = |\mathbf{V}_{tot}| = |\mathbf{v}_{rel}| + \sum_{j=1}^{N} \Gamma_j \mathbf{V}_{ji}$$

(4.5)

In the above expressions, $\mathbf{v}_{ij}$ is the normalized velocity induced at section $i$ by horseshoe
vortex \( j \), calculated as

\[
v_{ij} = \frac{1}{4\pi} \left[ \delta_{ij} \left( \frac{r_{ij}}{r_{ij}^2} \frac{r_{ij} \times r_{ij}}{r_{ij}^2} \right) + \frac{u_{\infty} \times r_{ij}}{r_{ij}^2} - \frac{u_{\infty} \times r_{ij}}{r_{ij}^2} \right]
\]

where \( \delta_{ij} \) is the Kronecker delta (1 if \( i = j \), 0 if \( i \neq j \)). With Equations (4.6) and (4.7), Equation (4.4) defines a system of equations that can be solved for the horseshoe vortex strength, \( \Gamma_i \), at each wing section. Once the horseshoe vortex strengths are known, the forces and moments acting on the system of wings can be found using the 3D vortex lifting law. Additionally, a correction for the viscous drag can be added to this model based on 2D airfoil drag behavior\(^{20}\).

### 4.4 Incorporation of prop wash into numerical lifting line model

As previously stated in the derivation of the numerical lifting line method, the local upstream velocity, \( V_{\text{rel}} \), differs from the freestream velocity in that it can incorporate velocity induced by prop wash or rotation of the lifting surface. Thus, in order to incorporate the effects of prop wash on a lifting surface in question, it should be a simple matter of adding the velocity of the prop wash to the local upstream velocity vector, \( V_{\text{rel}} \), for any wing sections immersed in the prop wash. Unfortunately, this is not the case.

Previous studies using both numerical lifting line theory and panel methods have found that incorporating the prop wash velocities in this way tends to significantly over predict the influence of the prop wash on a wing when compared to experimental data\(^{1-3}\). Various reasons for this over prediction have been proposed, but none of these reasons can fully explain the difference observed. However, the application of a number of reduction factors to the prop wash velocities has been shown to provide good results over a range of experimental data.
In order to obtain accurate results, two reduction factors are applied: an Axial Reduction Factor (ARF) and a Swirl Reduction Factor (SRF). These reduction factors are applied to the axial and tangential prop wash velocities respectively, as previously calculated using the turbulent prop wash model, before they are added to the local upstream velocity vectors, \( V_{rel} \), of the lifting line model. This is given as

\[
V_{rel_i} = V_\infty + \sum_k \left[ (1 - ARF) V_{i, a_k} + ((1 - SRF) + V_{i, t_k} \right]
\]

where \( N_p \) is the number of propellers in the system, \( a_k \) is the unit vector in the direction of the propeller axis, and \( t_k \) is the unit vector in the direction of the tangential velocity. It has been found that reduction factors of \( ARF = 1 \) and \( SRF = 0.6 \) yield good results when compared to the experimental data available. This means that none of the axial velocity and only 40% of the tangential velocity from the prop wash is applied to the lifting line model. The effect of these reduction factors on the lift distribution is shown in the next section.

4.5 Results of complete propeller-wing model

The results of the complete propeller-wing model were compared against experimental wind tunnel results from a number of sources. It should be noted that each source provided a different level of information detailing the geometry and operating condition of their test setups. As such, an effort has been made to create a numerical model that matches as closely as possible to the information provided. Where information was lacking, reasonable assumptions were made in an effort to make the numerical model as realistic as possible.
4.5.1 Comparison to results from Epema

The first comparison is made against results gathered by Epema at the Delft University of Technology\(^2\). In this test, a half wing with a propeller in tractor configuration mounted along its semispan was combined with a wall at its root to model a twin-engine general aviation aircraft. As this model is quite intricate, the reader is referenced to the thesis by Epema for a full description of the geometry\(^2\). As the description of the geometry is quite complete, the numerical model was able to use all of the parameters specified by Epema. The test was run at a freestream velocity of 19 m/s, a wing angle of attack of 4 degrees, and a propeller advance ratio of 0.695.

Figure 4.2 shows the variation in the normalized lift coefficient across the semispan of the wing model. Note in the experimental results that the propeller causes an increase in the lift of the wing on the up-rotating side and a corresponding decrease in the lift on the down-rotating side of the propeller. It can also be clearly seen that the direct application of the prop wash velocities to the numerical lifting line model (NLLM) without the reduction factors results in a strong over prediction of the effect of the propeller on the lift distribution. Additionally, application of only the ARF or SRF also results in an over prediction of the propeller effects.

Figure 4.3 shows a comparison of the experimental results to the numerical results obtained from the NLLM with both ARF = 1.0 and SRF = 0.6. As can be seen, application of these reduction factors results in a lift distribution that closely matches experimental results. The exact reason why these reduction factors are necessary is unknown. Further work could be done to investigate these reduction factors and to fine tune their values to provide better results when compared to a wider range of experimental results.
Fig. 4.2 Lift distribution across wing semispan with varied application of prop wash reduction factors.

Fig. 4.3 Lift distribution from Epema vs. NLLM with ARF=1 & SRF=0.6.
4.5.2 Comparison to results from Veldhuis

The next comparison is made against results gathered by Veldhuis at the Delft University of Technology\textsuperscript{1}. His wind tunnel model, denoted PROWIM, consists of a straight wing of aspect ratio $AR = 5.33$ with no twist, constant chord and airfoil section NACA 64$^2$-A015. Its half span is 0.64 m. A 4-bladed propeller of 0.236 m diameter is attached to the wing with a nacelle. It should be noted that the exact geometry of the propeller was not specified, so the propeller was modeled with the same chord distribution and airfoils as the Epema\textsuperscript{2} propeller, a pitch ratio of 1.4, and a pitch increment of -5.7 degrees. The test was run at angles of attack of 0, 4, and 10 degrees, and a propeller advance ratio of 0.85.

Figure 4.4 shows a comparison of the lift distributions obtained from the wind-tunnel test and the numerical model. As can be seen, the level of agreement between experimental and numerical results is good, but not quite as good as the comparison with the results from Epema. This is likely due to the fact that the geometry of the propeller in the PROWIM model was not well described. Therefore, despite best efforts, it is likely that the numerical model does not accurately model the aerodynamics of the propeller. This discrepancy then becomes evident in the disagreement in the lift distributions.

4.5.3 Comparison to results from Stuper

The final comparison is made against results gathered by Stuper\textsuperscript{17}. His wind tunnel model consisted of a straight wing with a span of 80 cm, a chord of 20 cm, and a symmetric airfoil section, Göttingen 409. This wing is placed between two circular end disks with a diameter of 32 cm. In order to numerically simulate these end disks, straight wing sections with a 32 cm span and chord were placed at the wingtips of the NLLM model.
A two-bladed propeller with a diameter of 15 cm and a pitch ratio of 0.4 is suspended in front of the wing model. The chord distribution is specified by Stuper, but the airfoil geometry of the propeller is not well specified. Therefore, the following airfoil parameters were used:

\[
\alpha_L = 0, \quad \tC_{L,\alpha} = 6.1314, \quad \tC_{L,max} = 1.4, \quad \tC_{D0} = 0.0079, \quad \tC_{DL} = -0.00085, \quad \text{and} \quad \tC_{DL2} = 0.01714
\]

The test was run at angles of attack of 4 and 8 degrees, and a propeller advance ratio of 0.15. However, running the numerical model at an advance ratio of 0.15 produces results that over predict the dynamic pressure within the slipstream and correspondingly over predict the effect of the prop wash on the wing, even with the ARF and SRF. It is believed that the cause of this discrepancy is a misunderstanding of the advance ratio. In his report, Stuper never defines the variables used in calculating the advance ratio. It is possible that he used the propeller radius instead of the diameter as a length scale, which
would make his advance ratio correspond to a traditionally defined advance ratio of 0.3. When the numerical analysis is run with an advance ratio of 0.3, the results for the dynamic pressure in the prop wash and the lift distribution match much more closely. The lift distribution across the wing can be seen in Figure 4.5.

Fig. 4.5 Lift distribution from Stuper vs. NLLM with ARF=1 & SRF=0.6.

4.6 Grid resolution study of combined model

The convergence of the combined propeller-wing model was tested by varying the spanwise nodes of a straight wing with a wingspan of 2 m, chord of 0.5 m, and a NACA 2412 airfoil. This wing had the propeller from the previous grid resolution studies set 0.75 meters in front of its leading edge at the root. The freestream velocity was set at 10 m/s and the total lift and rolling moment coefficients were calculated. These values were chosen as they can be strongly effected by immersing the wing in prop wash. Figure 4.6 shows the variation in the lift and rolling moment coefficients as a function of NLLM grid resolution.
The percent error in these values was calculated using the value with the highest resolution as the reference point. These results can be seen in Figure 4.7. From this analysis, the convergence of the combined propeller-wing model is determined to have an order of about 2–2.25. This order of convergence may vary slightly depending on the geometry and operating conditions. Based on this analysis, it is recommended that a grid resolution of 40 be used in the NLLM portion of the combined model as it provides results with less than 0.05% error while maintaining good computational efficiency.
Fig. 4.7 Percent error in $C_L$ & $C_\ell$ as function of spanwise nodes.
CHAPTER 5

CONCLUSION

Propellers and the prop wash they create can have a strong effect on the aerodynamics of many aircraft. As such, it is important that the effects of propellers on lifting surfaces be well understood and effectively modeled. There are many different methods for modeling these effects, ranging from high-order methods, such as CFD, to low-order methods, such as panel or lifting line methods. Low-order methods are desirable for a wide range of applications because of their ability to provide results of reasonable accuracy at a low computational cost. These applications include such things as initial design, optimization, and flight simulation. Because of this, an improved low-order method for modeling propeller-wing interactions has been developed and shown to provide results of better accuracy than existing methods while being computationally efficient.

The proposed low-order method incorporates a propeller blade element model, a novel turbulent prop wash model, and a numerical lifting line model. The blade element model and the turbulent prop wash model have been compared to experimental results and shown to provide good accuracy. The final model for propeller-wing interaction has also been shown to provide results with good accuracy by combining these computationally efficient, low-order models.

5.1 Propeller blade element model

A numerical model based on propeller blade element theory has been employed to model the aerodynamics of the propeller. This model calculates the forces and moments produced by the propeller by analyzing the aerodynamics of individual sections along the propeller blade. Doing so also allows for the calculation of the induced velocities in the
plane of the propeller, which are essential for the subsequent prop wash model. This model is further expanded to incorporate factors such as post-stall airfoil properties and rotational stall delay effects.

Although this propeller model requires that several simplifying assumptions be made, it is shown to provide accurate results over a broad range of operating conditions and propeller geometries. Incorporation of factors such as post-stall airfoil properties and rotational stall delay effects helps to improve the accuracy of the results at certain operating conditions, namely operation at low advance ratios. The results of this model are compared with experimental measurements obtained by the University of Illinois and are shown to provide good agreement. However, the accuracy of the results depends strongly on an accurate characterization of the propeller geometry within the model, namely pitch and chord distributions and airfoil properties.

5.2 Turbulent Prop Wash Model

There are a number of existing methods for modeling the development of the prop wash. Unfortunately, the majority of these methods either ignore the effects of turbulence or model them for only a limited range of applicable cases. Thus, a novel prop wash model was developed that models the effects of turbulence using observations drawn from the development of turbulent jets.

This turbulent prop wash model is shown to provide results of reasonable accuracy over a wide range of operating conditions and propeller geometries. Although the level of accuracy can vary from case to case, the true strength of this model lies in its ability to model the effects of turbulence. For certain operating conditions and geometries, including propellers operating at a low advance ratio or propellers with lifting surfaces far
downstream in their prop wash, the effects of turbulence are very apparent. In these cases, modeling the propeller slipstream as purely inviscid can result in a large over prediction of the prop wash velocities and a significant under prediction of the radius of the prop wash. Thus, although this turbulent prop wash model is not perfect, the results it provides are significantly more accurate than those provided by a simple inviscid prop wash model.

5.3 Numerical Lifting Line Model

A modern, numerical adaptation of Prandtl’s lifting line theory, as developed by Phillips, is used to model the aerodynamics of the lifting surfaces. This numerical lifting line model (NLLM) models the lifting surfaces of an aircraft as a series of horseshoe vortices bound to the quarter-chord of the wing and aligned with the freestream. The circulation of these vortices is related to the 2D section airfoil properties and a system of equations is created and solved to determine the inviscid forces and moments produced by the lifting surfaces. This NLLM is used because it provides accurate results at a fraction of the computational cost of inviscid panel methods or computational fluid dynamics\textsuperscript{16}.

To calculate the influence of the prop wash on the lifting surfaces, the prop wash velocities are added to the local section velocity for any sections of the lifting surfaces immersed in the prop wash. However, it has been shown that directly imposing all of the prop wash velocity on the local section velocity results in a significant over prediction of the influence of the prop wash\textsuperscript{1–3}. Thus, an Axial Reduction Factor (ARF) and a Swirl Reduction Factor (SRF) are applied to the prop wash velocity to reduce the impact of the prop wash on the lifting surfaces. These reduction factors are determined to have values of 1 and 0.6 respectively, meaning that none of the axial velocity and only 40\% of the tangential velocity is applied to the lifting surface. Application of these factors causes the
numerical results to align much more closely with experimental results from a number of sources.

5.4 Recommendations for Future Work

It is recommended that future work be done to improve the turbulent prop wash model and the influence of the propwash on the lifting surfaces. A number of assumptions were made and semi-empirical correction factors were added to the prop wash model to improve the agreement of its results with experimental data. Further work could be done to improve this model and its agreement without the incorporation of so many correction factors.

Regarding the influence of the prop wash on the lifting surfaces, the exact reason why the reduction factors are necessary is not well understood. Further work could be done to gain a better understanding of these reduction factors and to fine-tune their values to provide the best agreement with a broader range of experimental data.

5.5 Resulting Program: MachUp_Py

The improved low-order model presented in this work has been implemented in MachUp_Py, a Python adaptation of the open-source aerodynamic analysis tool developed by the Aerolab at Utah State University. This program is accessible via an web-based, graphical interface for basic applications or via a python library for more advanced applications.
REFERENCES


26 Thumthae, C., and Chitsomboon, T., “Improved 3D stall delay model for wind turbine design,” 2013, p. 11.


Prandtl, L., “Uber Tragflugel kleinsten induzierten Widerstandes.”