Aerodynamic interaction effects present in tiltable over-the-wing propeller systems

A numerical study

Sören Adrian Bölk

TU Delft
Aerodynamic interaction
effects present in tiltable
over-the-wing propeller
systems

A numerical study

by

Sören Adrian Bölk

to obtain the degree of Master of Science
at the Delft University of Technology,
to be defended publicly on Monday February 24, 2020 at 13:30.

Student number: 4712900
Thesis committee: R. de Vries M.Sc., TU Delft
                  Prof. dr. ir. L.L.M. Veldhuis, TU Delft, supervisor
                  Prof. dr. ing. G. Eitelberg, TU Delft
                  Dr. ir. B.W. van Oudheusden, TU Delft

An electronic version of this thesis is available at http://repository.tudelft.nl/.
## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Summary</td>
<td>v</td>
</tr>
<tr>
<td>List of Figures</td>
<td>vii</td>
</tr>
<tr>
<td>List of Tables</td>
<td>xi</td>
</tr>
<tr>
<td>Nomenclature</td>
<td>xiii</td>
</tr>
<tr>
<td>I Background</td>
<td>1</td>
</tr>
<tr>
<td>1 Introduction</td>
<td>3</td>
</tr>
<tr>
<td>1.1 Research Objective</td>
<td>4</td>
</tr>
<tr>
<td>1.2 Thesis outline</td>
<td>5</td>
</tr>
<tr>
<td>2 Theory and Literature Review</td>
<td>7</td>
</tr>
<tr>
<td>2.1 Propeller aerodynamics</td>
<td>7</td>
</tr>
<tr>
<td>2.2 Aerodynamic effects of the wing on the propeller performance</td>
<td>10</td>
</tr>
<tr>
<td>2.3 Aerodynamic effects of the propeller on the wing performance</td>
<td>11</td>
</tr>
<tr>
<td>2.4 Propeller–boundary-layer interaction</td>
<td>14</td>
</tr>
<tr>
<td>2.5 Computational methods for propeller interaction studies.</td>
<td>16</td>
</tr>
<tr>
<td>II Methodology</td>
<td>19</td>
</tr>
<tr>
<td>3 Set-up</td>
<td>21</td>
</tr>
<tr>
<td>3.1 Experimental set-up</td>
<td>21</td>
</tr>
<tr>
<td>3.1.1 Wind Tunnel Facility and Model</td>
<td>21</td>
</tr>
<tr>
<td>3.1.2 Measurement Techniques</td>
<td>22</td>
</tr>
<tr>
<td>3.2 Numerical set-up</td>
<td>24</td>
</tr>
<tr>
<td>3.2.1 Governing equations</td>
<td>24</td>
</tr>
<tr>
<td>3.2.2 Turbulence modelling</td>
<td>26</td>
</tr>
<tr>
<td>3.2.3 The overset approach</td>
<td>26</td>
</tr>
<tr>
<td>3.2.4 Computational domain and boundary conditions</td>
<td>29</td>
</tr>
<tr>
<td>4 Verification</td>
<td>33</td>
</tr>
<tr>
<td>4.1 Error estimation and accuracy</td>
<td>33</td>
</tr>
<tr>
<td>4.1.1 Isolated Wing</td>
<td>34</td>
</tr>
<tr>
<td>4.1.2 Isolated propeller</td>
<td>36</td>
</tr>
<tr>
<td>4.2 Influence of overset modelling</td>
<td>37</td>
</tr>
<tr>
<td>5 Validation</td>
<td>41</td>
</tr>
<tr>
<td>5.1 Isolated wing boundary-layer profile</td>
<td>41</td>
</tr>
<tr>
<td>5.2 Isolated wing pressure distribution</td>
<td>42</td>
</tr>
<tr>
<td>5.3 Flow field over the flap of the wing</td>
<td>43</td>
</tr>
<tr>
<td>III Results</td>
<td>45</td>
</tr>
<tr>
<td>6 Aerodynamic characteristics of individual components</td>
<td>47</td>
</tr>
<tr>
<td>6.1 Isolated wing</td>
<td>47</td>
</tr>
<tr>
<td>6.2 Isolated propeller</td>
<td>49</td>
</tr>
</tbody>
</table>
7 Baseline over-the-wing configuration 51
   7.1 Comparison to experimental results ........................................... 51
   7.2 Influence of the wind tunnel walls ........................................... 58
      7.2.1 Isolated wing ................................................................. 58
      7.2.2 Wing-propeller system ..................................................... 62
   7.3 Aerodynamic interaction phenomena .......................................... 64
      7.3.1 Propeller-induced pressure and velocity fields ......................... 64
      7.3.2 Flow separation on the wing ............................................. 68
   7.4 Propeller loading distributions .............................................. 71
   7.5 Wing loads ................................................................. 73

8 Inclined over-the-wing configuration 75
   8.1 Aerodynamic interaction phenomena .......................................... 76
      8.1.1 Propeller-induced pressure and velocity fields ......................... 76
      8.1.2 Flow separation on the wing ............................................. 79
   8.2 Propeller loading distributions .............................................. 80
   8.3 Wing loads ................................................................. 82
   8.4 Comparison between baseline and inclined system performance ............ 84

IV Conclusions and recommendations 87

9 Conclusions 89

10 Recommendations 91

V Additional results 93
   A Isolated propeller performance .............................................. 95
   B Overset influence on velocity gradients .................................... 96
   C Vortex shedding behind wing in the wind tunnel ......................... 98
   D Baseline thrust distribution with wind tunnel walls ...................... 100
   E Change in skin friction on the baseline wing ............................ 101
   F Streamlines on the wing with inclined propeller ......................... 102

Bibliography 103
Summary

To limit the ecological impact of aviation while serving the increasing demand for air travel by a world-population growing in size and wealth, new power and propulsion concepts for aircraft are developed. One of such advanced propulsion concepts, intending to reduce fuel consumption and noise emission, is distributing over-the-wing (OTW) propellers along the wingspan to exploit advantageous aerodynamic interaction effects and benefit from the improved propulsive efficiency of propellers compared to turbofans.

Previous experimental and numerical research shows that an OTW propeller alters the pressure distribution of the wing such that decreased pressure can be found in front of the propeller and increased pressure behind it. With the propeller close to the trailing edge of the wing, this can lead to a significant lift increase due to a large area of enhanced suction in front of the propeller, and pressure drag decrease due to leading-edge suction and a reduced backward-facing suction peak on the flap behind the propeller. However, the velocities above the wing cause a non-uniform inflow to the propeller featuring increased velocities and a change in the direction of the velocities compared to a uniform axial inflow. Inconsistencies exist in the literature regarding how the thrust of an OTW propeller is affected by the non-uniform inflow above a wing since the increased inflow speed typically reduces thrust while a slight deviation in the inflow direction from a perfectly axial inflow can increase the thrust. Furthermore, recent studies show that an OTW propeller can cause flow separation on a subsequent flap which is attributed to the interaction of the slipstream with the wing, and the blade tip vortices with the boundary-layer of the wing. However, a study on the influence of the blade tip vortices on flow separation on a flap due to their interaction with the boundary-layer of the wing is missing in the literature. This is because many numerical aerodynamic investigations of OTW propeller applications were conducted either inviscid or as steady RANS simulations with an actuator disc for reduced computational cost. Therefore, the objective of this thesis is to contribute to the development of distributed OTW propeller systems by studying the aerodynamic interaction between a propeller above a deflected flap and the wing, through unsteady RANS simulations. The goal thereby is to understand the aerodynamic interaction of the blade tip vortices with the boundary-layer. Furthermore, the potential of inclining the propeller to prevent flow separation will be investigated.

To capture the blade tip vortices, a rotating overset mesh featuring a full-blade propeller model is used. First, the propeller is placed above the hinge line of the deflected flap of a simplified wing model at zero degrees angle of attack, with the propeller axis horizontally aligned with the chord-line. The numerical results of this baseline configuration are compared to experimental results for validation purposes. Furthermore, the numerical results are used to study the effect of the wind tunnel walls on the aerodynamic interaction effects. The most significant difference between the numerical and experimental results is stronger flow separation over the flap in the experiments than in the CFD simulations due to an underestimation of the blade tip vorticity in the numerical results caused by numerical diffusion. Moreover, the overset interface contributes to an inaccurate prediction of flow separation by altering the flow gradients in the boundary-layer. Combined, these effects lead to a $\Delta C_{l,p} = 0.052$ higher pressure-lift coefficient and $\Delta C_{d,p} = 0.0038$ lower pressure-drag coefficient in the numerical results, which become relatively large compared to the low lift coefficient ($C_l = 0.1$) and pressure-drag coefficient ($C_{d,p} = 0.008$) of the isolated wing. Furthermore, the wind tunnel walls increase the propeller-induced drag reduction due to an additional decrease in suction over the flap caused by slipstream blockage. The propeller-induced lift increase and flow separation are not significantly affected by the wind tunnel walls.

Flow separation over the flap in the baseline configuration is caused by a momentum deficit between the flap and the contracting slipstream, as well as the additional adverse pressure gradient created by the blade tip vortices. To prevent flow separation, the propeller can be inclined with the flap as if it were physically attached to it and, thus, increases the momentum in the boundary-layer of the flap by blowing high momentum flow over it. However, the non-uniform inflow to the inclined propeller disc causes a decrease in thrust close to the wing. In the baseline case, on the other hand, vertically induced velocities by the wing increasing the blade angle of attack for the up-going blade dominate over the axial velocity increase induced by the wing, such that the propeller exhibits no loss in thrust compared to the isolated case. This results in a total thrust of the baseline propeller that is comparable to the isolated propeller thrust and a reduced propeller thrust in the inclined position compared to both the baseline and isolated propeller. The reduced thrust of the inclined
propeller close to the wing, results in less low-pressure induced in front of it, hence, less lift of the wing than in the baseline configuration. The postponed flow separation and higher pressures over the flap in the inclined configuration, however, reduce the pressure drag of the wing more than in the baseline case.

Besides demonstrating the necessity of accurately capturing the blade tip vortices to obtain useful numerical results of an OTW propeller application, this thesis provides a design solution to prevent flow separation over a deflected flap by inclining the propeller. This contributes to the development process of distributed OTW propeller systems by highlighting the importance of accurately capturing the blade tip vortices, giving recommendations on how to model an OTW propeller application with an overset interface, and motivating further research as the baseline configuration showed an improved propulsive efficiency. With this in mind, the wing of an OTW propeller system needs to be designed such that performance penalties on the propeller thrust are mitigated, for instance by enforcing flow deceleration in front of the propeller. Alternatively, the vectored thrust of the inclined propeller can be used to compensate for the reduced lift of the wing compared to the baseline configuration.
List of Figures

2.1 Velocities at a blade element. Figure adapted from Ref. [75]. ...................................................... 8
2.2 Control surface for the momentum integral, from Ref. [75]. ............................................................... 8
2.3 Axial velocity $v_x$ increase and both total $p_t$ and static $p_s$ pressure development across the actuator disc. The pressures are not to scale. Figure adapted from Ref. [72]. .................................................. 9
2.4 Distributions of effective blade angle of attack and relative local thrust at $\alpha = 0$, from Ref. [46]. .... 11
2.5 OTW propeller configurations for cruise (left) and high lift (right), from Ref. [40]. ............................... 12
2.6 Wing lift and pressure drag increase with respect to the propeller-off conditions in high-lift configuration, from Ref. [40]. ................................................................. 13
2.7 Steady (left) and unsteady (right) simulation of the flow field around a channel wing in a vertical plane through the propeller axis, from Ref. [48]. ................................................................. 13
2.8 Unsteady pressure distributions of the channel wing below the propeller axis, from Ref. [48]. .... 14
2.9 Instantaneous velocity flow field for high thrust ($J = 0.58$) 10 mm away from the wall. The red circles indicate the counter rotating vortices. Figure from Ref. [83]. .................................................. 15
2.10 Mean velocity contours and vectors through the tip gap for moderate thrust $J = 1.05$ (top) and high thrust $J = 0.58$ (bottom), from Ref. [51]. The black lines project the rotor on the wall in the $z$-$x$-plane. .................. 16

3.1 Rear view of the wing and the propeller inside the test section including annotations for the individual components. Figure adapted from Ref. [19]. ................................................................. 21
3.2 Sketch of the wing profile and a close-up of the blade tip region. ..................................................... 22
3.3 Image indicating the position of the pressure taps, microphones (not used in this study), and the slit for the PIV laser sheet. Figure adapted from Ref. [19]. .................................................. 23
3.4 Rear view of the flap and propeller indicating the laser sheet and the field-of-view (FOV) of the PIV measurements. Figure adapted from Ref. [19]. ................................................................. 24
3.5 Component mesh of a cylinder on a Cartesian background mesh, from Ref. [59]. ............................... 26
3.6 Wing (background) mesh and propeller (component) mesh before hole cutting. ............................ 27
3.7 Wing (background) mesh and propeller (component) mesh after hole cutting. ............................ 27
3.8 Overlap minimisation based on matching cell volumes. ................................................................. 28
3.9 Overlap minimisation based on boundary distance. ................................................................. 28
3.10 Overset interface with a higher priority for the propeller mesh. ..................................................... 28
3.11 Donor search in the overlap area between the wing surface and a blade tip. ................................ 29
3.12 Computational domain of the isolated propeller. ................................................................. 30
3.13 Computational domain of the baseline configuration with wind tunnel walls. ............................. 30
3.14 Computational domain of the inclined configuration without wind tunnel walls. ...................... 31
3.15 Position of the inclined over-the-wing propeller. ................................................................. 31

4.1 Lift coefficient per iteration. ................................................................. 34
4.2 Drag coefficient per iteration. ................................................................. 34
4.3 2D lift coefficient refinement. ................................................................. 35
4.4 2D drag coefficient refinement. ................................................................. 35
4.5 Non-uniform axial velocity distribution along a vertical line at the chordwise propeller location. .... 35
4.6 Thrust coefficient per iteration. ................................................................. 36
4.7 Torque coefficient per iteration. ................................................................. 36
4.8 Thrust coefficient refinement. ................................................................. 36
4.9 Torque coefficient refinement. ................................................................. 36
4.10 Thrust distribution. ................................................................. 37
4.11 Axial velocity through the tip vortices at a distance of $c/R = 0.0738$ to the propeller tip. .... 37
4.12 Hybrid overset mesh of the baseline propeller above the flap. ................................................................. 37
4.13 Boundary layer profile of the wing at the propeller location and pressure distribution at the half span of the wing compared between the isolated wing without an empty propeller domain, and the wing with an empty, non-rotating, overset domain. ................................................... 38
4.14 Influence of the overset interface on the shear stresses on the wing. ................................................... 39

5.1 2D boundary layer profiles for different $k-\omega$ turbulence model options. ................................. 42
5.2 Pressure coefficient distribution along the airfoil chord for different $k-\omega$ turbulence model options. ........................................................................................................................................... 42
5.3 Comparison between the CFD RANS results and the PIV measurement for the velocity component in the x direction over the flap. ........................................................................................................................................... 43
5.4 Axial velocity profile at the axial coordinate $x/c = 0.9$. Errorbars indicate the uncertainty in the velocity obtained from PIV. ........................................................................................................................................... 43
5.5 Comparison between the CFD RANS results and the PIV measurement for the turbulent kinetic energy of the flow over the flap. ........................................................................................................................................... 44

6.1 Axial velocity and streamlines around the isolated wing. ................................................... 48
6.2 Sketch of the angle between the streamlines and the baseline, and inclined, propeller planes. 48
6.3 Change in inflow conditions .................................................................................................................... 49
6.4 Pressure and skin friction distributions of the wing profile at $y/c = 0$ for the two nacelle configurations. ........................................................................................................................................... 49
6.5 Total pressure in the mid-plane. Contour lines of $C_{\omega,z}$ indicate the blade tip vortices. The dashed lines are the survey locations plotted in Fig. 6.6. ........................................................................................................................................... 50
6.6 Axial velocity and pressure coefficient along horizontal lines at increasing distance from the isolated propeller tip. ........................................................................................................................................... 50

7.1 Convergence plot for the lift and drag coefficient (a) and the axial velocity profile at one radius ahead of the propeller (b) (baseline configuration with wind tunnel walls). ................................................... 52
7.2 Drawing of the wing profile, propeller cross-section, and the vertical line from which the axial velocity profiles in Fig.7.1(b) were extracted. ........................................................................................................................................... 52
7.3 Contour of the time-averaged pressure on the suction side of the wing with the propeller running. The zig-zag lines connect the discrete points at which pressure data was extracted for the comparison to the wind tunnel experiments. ........................................................................................................................................... 53
7.4 Comparison between the pressure deltas from the traversing pressure measurement in the experiment (a), and CFD (b). Pressures from CFD were extracted at the same points as in the experiments. The colourbars are not saturated for improved comparability. ........................................................................................................................................... 54
7.5 Pressure difference between propeller-on and propeller-off on the suction side of the wing at $y/c = 0$. The axial coordinate $x'$ has its origin at the propeller location. ........................................................................................................................................... 54
7.6 Comparison between the time-averaged CFD RANS results and the steady PIV measurement for the velocity component in the x direction over the flap. The pink line connects points of zero axial velocity. ........................................................................................................................................... 55
7.7 Velocity profiles over the flap at three axial coordinates. ........................................................................................................................................... 55
7.8 Comparison between the instantaneous CFD RANS results and the phase-locked PIV measurement for the out-of-plane vorticity over the flap. ........................................................................................................................................... 56
7.9 Pressure distribution of the wing-section at $y/c = 0$, with and without influence of the propeller, from the experiment and CFD. Errorbars have been added to every second measurement point on the suction side of the wing. ........................................................................................................................................... 57
7.10 Convergence plot for the thrust coefficient (a) for all blades combined and for one blade individually, and thrust distributions (b) of the same blade after different amounts of propeller revolutions have passed. ........................................................................................................................................... 57
7.11 Performance curve of the installed propeller in the experiment including the thrust coefficient from CFD (red circle) for comparison. ........................................................................................................................................... 58
7.12 Pressure distribution of the 2D airfoil. ........................................................................................................................................... 59
7.13 Inflow velocity profile at the chordwise propeller location (propeller not present). 59
7.14 Lift distribution for different wall refinement. ........................................................................................................................................... 60
7.15 Visualisation of the junction flow at the intersection between the wing and wind tunnel wall. ........................................................................................................................................... 61
7.16 Convergence plot for the lift, drag, thrust of all blades, and thrust of blade 1 (baseline configuration without wind tunnel walls). ........................................................................................................................................... 62
7.17 Spanwise lift and drag coefficients $C_l$ and $C_d$ averaged over 60 degrees of propeller rotation for the cases with and without wind tunnel walls. 63
7.18 Pressure distribution of the wing profile at $y/b = 0$ averaged over 60 degrees of propeller rotation for the cases with and without wind tunnel walls. 63
7.19 Axial component of the pressure coefficient for the wing profile at $y/c = 0$. 63
7.20 Boundary layer over the flap at $x/c = 0.9$, extracted along the survey line in Figure 7.21. The velocity $u$ is parallel to the flap’s surface, positive towards the trailing edge. 64
7.21 Contour of the axial velocity distribution over the flap with wind tunnel walls. The black line indicates where a boundary layer profile is extracted. 64
7.22 Phase averaged axial velocity and streamlines in the mid-plane at $y/c = 0$ (a). Phase averaged induced pressure and velocities in the mid-plane at $y/c = 0$ (b). 65
7.23 Out-of-plane vorticity $C_{w,y}$ and propeller-induced velocities in the $y$-plane at $y/c = 0$ for $\phi = 2^\circ$. 66
7.24 Velocity profile parallel to the flap wall at $x/c = 0.9$, extracted along the black line in Figure 7.23. 66
7.25 Vorticity magnitude $C_w$ and propeller-induced velocities in a $z$-plane, parallel to the wing’s suction side wall, at the mid-height of the tip gap. 66
7.26 Time-averaged induced static pressure on the wing’s suction side. The propeller blades are superimposed in grey. 67
7.27 Pressure distribution of the wing profile at $y/c = 0$. 67
7.28 Transient development of the pressure coefficient for the wing profile at $y/c = 0$. 68
7.29 Sectional drag coefficient of the wing profile at $y/b = 0$ at different blade positions. 68
7.30 Phase averaged skin friction coefficient on the wing’s suction side and wall shear-lines. The dashed line indicates the chordwise location of separation for the isolated wing. The colour bar does not span the full range of friction coefficient values to improve readability. 69
7.31 Boundary-layer profiles $0.5R$ ahead of the propeller at three different spanwise locations. 69
7.32 Time averaged skin friction coefficient on the wing, and velocities in the chordwise and spanwise direction in vertical planes at the spanwise locations $y/c = -0.075, 0,$ and $0.075$. The colourbars do not span the full range of values to improve readability. 69
7.33 Unsteady and averaged skin friction coefficient on the suction side of the wing profile at $y/c = 0$. In Figure (b) the averaged skin friction coefficient (Avg) is compared to the skin friction of the isolated wing without the propeller (Iso). 70
7.34 Velocities induced by the wing, in front of the down-going blade. 71
7.35 Thrust distribution on the propeller disc (a) and difference with respect to the isolated propeller (b). The range of the colourmap was defined for better comparability to the inclined propeller thrust distributions in Section 8.2. 72
7.36 Total pressure distribution in the disc plane. Image adapted from Ref. [40]. 72
7.37 Thrust distribution for different phase angles compared to the isolated blade. 72
7.38 Spanwise lift, drag, and lift-to-drag ratio of the wing, averaged over 60 degrees of propeller rotation. 73
7.39 Spanwise lift and drag of the wing for different propeller phase angles. 74
7.40 Lift and Drag deltas for wing segments of increasing span. The span $b'$ is normalised with the propeller diameter $D$. 74
8.1 Unsteady convergence plot for the lift, drag, thrust of all blades, and thrust of blade 1 (inclined and baseline configuration without wind tunnel walls). 75
8.2 Phase averaged axial velocity and streamlines in the mid-plane at $y/c = 0$ (a). Phase averaged induced pressure and velocities in the mid-plane at $y/c = 0$ (b). 76
8.3 Out-of-plane vorticity $C_{w,y}$ and propeller-induced velocities in the $y$-plane at $y/c = 0$ for $\phi = 2^\circ$. 77
8.4 Time-averaged boundary-layer over the flap at $x/c = 0.9$, below the propeller axis. The velocity $u$ is parallel to the propeller axis and extracted along the survey-line in Figure 8.3. 77
8.5 Averaged propeller-induced static pressure on the wing's suction side. 77
8.6 Pressure distribution of the wing profile at $y/c = 0$. 78
8.7 Transient development of the pressure coefficient for the wing profile at $y/c = 0$. 78
8.8 Sectional drag coefficient of the wing profile at $y/b = 0$ at different time instances, i.e. for different blade positions. 79
8.9 Averaged skin friction coefficient on the wing’s suction side and wall shear-lines. The propeller blades are shown as an example for a phase angle of $\phi = 30^\circ$. The colormap does not span the full range of friction coefficient values.

8.10 Unsteady and averaged skin friction coefficient on the suction side of the wing profile at $y/c = 0$. In Figure (b), the averaged skin friction coefficient is compared to the skin friction of the wing with nacelle (prop-off), and the baseline configuration.

8.11 Thrust distribution on the propeller disc (a) and difference with respect to the uninstalled case (b).

8.12 Thrust distribution for different phase angles compared to the isolated blade. The dashed lines represent the thrust distributions of the baseline propeller.

8.13 Sketch of the forces acting on the inclined propeller. The vectors are not to scale.

8.14 Spanwise lift, drag, and lift-to-drag ratio of the wing, averaged over 60 degrees of propeller rotation.

8.15 Spanwise lift and drag of the wing for different propeller phase angles.

8.16 Lift and Drag deltas for wing segments of increasing span.

A.1 Chord distribution and blade pitch angle of a propeller blade, taken from Ref. [70].

A.2 Isolated propeller performance curves obtained from numerical and experimental data in Ref. [70].

B.1 Vertical gradient of the axial velocity in the plane along the propeller axis, zoomed into the tip region.

B.2 Vertical gradient of the axial velocity in the plane along the propeller axis.

B.3 Vertical gradient of the axial velocity in the propeller plane, viewed from behind the propeller.

C.1 Vorticity in [1/s] and Q-Criterion in a plane at $y/c = 0$ for the baseline propeller-off and propeller-on case inside the wind tunnel. The colourbars do not span the full range of vorticity to improve readability.

D.1 Thrust distribution in the baseline configuration with wind tunnel walls.

E.1 Difference in skin friction on the wing between propeller on and propeller off.

F.1 Pressure coefficient and streamlines on the suction side of the wing.
List of Tables

2.1 Estimated CPU time of various two-equation models for a fixed number of iterations relative to the CPU time of the Spalart-Allmaras model, from Ref [11]. .................................................. 18

3.1 PIV set-up from Ref. [19] ................................................................. 24

4.1 Error and uncertainty of the lift and drag coefficients in the units of the respective coefficients. 35

4.2 Error and uncertainty of the thrust and torque coefficients in the units of the respective coefficients. .......................................................... 36

4.3 Boundary-layer at \((x/c = 0.79, \ y/c = 0)\), and sectional lift and drag coefficients at \(y/c = 0\) with and without an empty overset domain. .................................................... 38

5.1 Boundary layer thicknesses for different \(k - \omega\) turbulence model options. ......................... 41

5.2 Lift coefficients for different \(k - \omega\) turbulence model options. ......................................... 42

7.1 Sectional pressure-lift and pressure-drag coefficients of the wing profile at \(y/c = 0\), for the propeller-off and propeller-on case. .................................................. 56

7.2 Sectional lift and drag coefficients from the 3D and 2D results of the isolated wing, with and without wind tunnel walls. .................................................. 60

7.3 Sectional lift and drag coefficients of the wing profile at \(y/c = 0\), for the propeller-off and propeller-on case, with and without walls. .................................................. 64

8.1 Table comparing the forces acting on the baseline and inclined propeller. Data was taken from zero and 30 degrees phase angle and the mean was calculated to average out the thrust fluctuation which have a wavelength of 60 degrees phase angle. ................................. 82

8.2 Balance of forces in z-direction for a wing-segment span of \(b'/D = 1.1\) and relative change with respect to the baseline case. ................................................. 84

8.3 Balance of forces in x-direction for a wing-segment span of \(b'/D = 1.1\) and relative change with respect to the baseline case. The thrust is positive in the negative x-direction. ............................................... 84

8.4 Table comparing the propulsive efficiency. ................................................................. 85
# Nomenclature

## Latin symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Test-section cross-sectional area</td>
<td>$[m^2]$</td>
</tr>
<tr>
<td>$b$</td>
<td>Wing span</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$b'$</td>
<td>Wing-segment span</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$c$</td>
<td>Wing chord</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$C_D$</td>
<td>Drag coefficient $C_D = \frac{D}{\frac{1}{2} \rho \infty c b}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$C_d$</td>
<td>Sectional drag coefficient $C_d = \frac{d}{\frac{1}{2} \rho \infty c}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$C_f$</td>
<td>Skin friction coefficient $C_f = \frac{f}{\frac{1}{2} \rho \infty}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$C_L$</td>
<td>Lift coefficient $C_L = \frac{L}{\frac{1}{2} \rho \infty c}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$C_l$</td>
<td>Sectional lift coefficient $C_l = \frac{l}{\frac{1}{2} \rho \infty c}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$C_P$</td>
<td>Power coefficient $C_P = \frac{P_s}{\rho n^2 D_p^5}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$C_T$</td>
<td>Thrust coefficient $C_T = \frac{T}{p n^2 D_p^4}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$C_T$</td>
<td>Thrust distribution $\frac{dC_T}{d(r/R)}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$C_Q$</td>
<td>Torque coefficient $C_Q = \frac{Q}{p n^2 D_p^5}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$C_{\omega}$</td>
<td>Vorticity coefficient $C_{\omega} = \frac{\omega}{2 \Omega}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$D$</td>
<td>Drag Force</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$D_p$</td>
<td>Propeller diameter</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$d$</td>
<td>Distance to a wall</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$d$ or $2D$ drag</td>
<td>Drag force</td>
<td>$[N/m]$</td>
</tr>
<tr>
<td>$F_x$</td>
<td>Component of the resultant force in the $x$-direction</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$F_z$</td>
<td>Component of the resultant force in the $z$-direction</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$h$</td>
<td>Wind tunnel height</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$h_i$</td>
<td>Representative grid cell size</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$J$</td>
<td>Advance ratio $J = \frac{V}{\pi D_p^2}$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy</td>
<td>$[m^2/s^2]$</td>
</tr>
<tr>
<td>$L$</td>
<td>Lift Force</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$l$</td>
<td>2D lift</td>
<td>$[N/m]$</td>
</tr>
<tr>
<td>$L/D$</td>
<td>Lift-to-drag ratio</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$N$</td>
<td>Propeller-normal force</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$n$</td>
<td>Rounds per seconds</td>
<td>$[1/s]$</td>
</tr>
<tr>
<td>$n$ or $2$ or $\text{number of cells}$</td>
<td>Rounds per seconds</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
<td>$[Pa]$</td>
</tr>
<tr>
<td>$p_s$</td>
<td>Shaft power</td>
<td>$[W]$</td>
</tr>
<tr>
<td>$q$</td>
<td>Dynamic pressure</td>
<td>$[Pa]$</td>
</tr>
<tr>
<td>$Q$</td>
<td>Propeller Torque</td>
<td>$[Nm]$</td>
</tr>
</tbody>
</table>

*xiii*
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R$</td>
<td>Propeller-resultant force</td>
<td>[N]</td>
</tr>
<tr>
<td>$r$</td>
<td>Propeller radius variable</td>
<td>[m]</td>
</tr>
<tr>
<td>$S$</td>
<td>Area</td>
<td>[m$^2$]</td>
</tr>
<tr>
<td>$T$</td>
<td>Propeller thrust</td>
<td>[N]</td>
</tr>
<tr>
<td>$T_c$</td>
<td>Thrust coefficient $T_c = \frac{T}{\rho u^3 S}$</td>
<td>[-]</td>
</tr>
<tr>
<td>$\hat{T}_c$</td>
<td>Thrust coefficient of wing-segment $\hat{T}_c = \frac{T}{\rho u \beta}$</td>
<td>[-]</td>
</tr>
<tr>
<td>$T_{x,\text{inst}}$</td>
<td>Installed thrust along the $x$-axis $T_{x,\text{inst}} =</td>
<td>F_x</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>[kg/m$^3$]</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Standard deviation</td>
<td>[-]</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>[kg/m$^3$]</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Standard deviation</td>
<td>[-]</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Skin shear stress</td>
<td>[N/m$^2$]</td>
</tr>
<tr>
<td>$\Phi$</td>
<td>Inflow angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Phase angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>Angular velocity</td>
<td>[rad/s]</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Vorticity</td>
<td>[1/s]</td>
</tr>
</tbody>
</table>

**Greek symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>Blade angle of attack</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Blade pitch angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\delta_{gh}$</td>
<td>Boundary-layer height</td>
<td>[m]</td>
</tr>
<tr>
<td>$\Delta(\cdot)$</td>
<td>Change due to the propeller $\Delta(\cdot) = (\cdot)^{\text{on}} - (\cdot)^{\text{off}}$</td>
<td>[-]</td>
</tr>
<tr>
<td>$\Delta(\cdot)$</td>
<td>Change due to the wind tunnel walls $\Delta(\cdot) = (\cdot)^{\text{WT}} - (\cdot)^{\text{FF}}$</td>
<td>[-]</td>
</tr>
<tr>
<td>$\Delta(\cdot)$</td>
<td>Change with respect to the wind tunnel experiments $\Delta(\cdot) = (\cdot)^{\text{num}} - (\cdot)^{\text{exp}}$</td>
<td>[-]</td>
</tr>
<tr>
<td>$\delta_f$</td>
<td>Flap deflection angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>Propeller tip gap</td>
<td>[m]</td>
</tr>
<tr>
<td>$\epsilon_{sb}$</td>
<td>Solid body blockage factor $\epsilon_{sb} = 0.74 \frac{W}{A^{3/2}}$</td>
<td>[-]</td>
</tr>
<tr>
<td>$\epsilon_{wb}$</td>
<td>Wake body blockage factor $\epsilon_{wb} = \frac{c}{\pi h} \left( \frac{c}{h} \right)^2$</td>
<td>[-]</td>
</tr>
<tr>
<td>$\eta_{\text{pro}}$</td>
<td>Propulsive efficiency $\eta_{\text{pro}} = \frac{TV}{P_s}$</td>
<td>[-]</td>
</tr>
</tbody>
</table>

**List of Tables**

- [List of Tables](#)
Sub- and superscripts

(\textsuperscript{\textit{exp}}) Experimental
(\textsuperscript{\textit{inst}}) Installed
(\textsuperscript{\textit{iso}}) Isolated
(\textsuperscript{\textit{num}}) Numerical
(\textsuperscript{\textit{off}}) Propeller-off
(\textsuperscript{\textit{on}}) Propeller-on
(\textsuperscript{\textit{N}}) Normal force direction
(\textsuperscript{\textit{p}}) Derived from pressure
(\textsuperscript{\textit{ref}}) Reference
(\textsuperscript{\textit{R}}) Resultant force direction
(\textsuperscript{\textit{s}}) Static
(\textsuperscript{\textit{t}}) Total
(\textsuperscript{\textit{u}}) Uncorrected
(\textsuperscript{\textit{x}}) Component in the \textit{x}-direction
(\textsuperscript{\textit{y}}) Component in the \textit{y}-direction
(\textsuperscript{\textit{z}}) Component in the \textit{z}-direction
(\textsuperscript{\textit{\infty}}) Free-stream value
(\textsuperscript{\textit{\prime}}) Coordinates in the propeller-centered coordinate system
(\textsuperscript{\textit{\cdot}}) Thrust coefficient based on the reference area of a wing-segment

Abbreviations

Avg  Time-averaged
BL   Boundary-layer
BSL  Baseline
CFD  Computational Fluid Dynamics
DES  Detached Eddy Simulations
DNS  Direct Numerical Simulation
EARSM Explicit Algebraic Reynolds Stress Model
EVM  Eddy Viscosity Model
FF   Far-field
IBF  Internally blown flap
LES  Large Eddy Simulation
LTT  Low-turbulence Tunnel
NS   Navier Stokes
OS   Overset
OTW  Over-the-wing
PIV  Particle Image Velocimetry
Prop Propeller
RANS Reynolds Averaged Navier Stokes
Re   Reynolds number
RSM  Reynolds Stress Model
SST  Shear Stress Transport
WT   Wind tunnel
Background
Introduction

Worldwide air travel is predicted to increase by 4.7% per year between the years 2008 and 2025 [15]. Due to this increasing demand for air travel, and consequently jet fuel, new solutions need to be found to limit the ecological impact of aviation while serving the demand for air travel by a population growing in size and wealth. With this in mind, the National Aeronautics and Space Administration (NASA) [8] and the European Commission [1] defined goals targeting emissions, noise, fuel burn, and field length. In order to meet these challenging requirements, new aircraft designs and new power sources, such as Hybrid-Electric Propulsion (HEP) [61], are developed, which in turn enable advantageous propulsion-integration concepts like Distributed Propulsion (DP) [54, 78]. DP is beneficial in terms of fuel savings and reduction of take-off noise as well as emissions [36]. However, at the current stage, these benefits are mainly realisable for small to medium range aircraft [27].

A possible way of reducing fuel consumption is using propellers instead of turbofans as they have a higher propulsive efficiency, and placing them close the wing to exploit beneficial aerodynamic interaction effects. Compared to the conventional tractor propeller configuration, for instance, mounting a propeller above a wing, in close proximity to it, can benefit the lift-to-drag ratio of the wing section beneath it by increasing the lift and reducing the pressure drag [16, 34] at both high-lift [47] and cruise [49] conditions. Furthermore, distributing over-the-wing propellers along the wingspan instead of mounting one large propeller can increase the wing loading at constant thrust allowing a reduced take-off length or wing size, which benefits the aviation infrastructure and the aircraft’s structural weight, respectively. Moreover, compared to tractor propellers, OTW propellers overcome ground clearance limitations and emit less noise to the ground due to shielding by the wing [21].

OTW propellers are not an entirely new concept, but were already tested in the first half of the 20th century [17, 53]. However, they are not used in present commercial aircraft designs, which might be attributed to the general stagnation in propeller research from the mid 50’s to the mid 70’s. In that period, low fuel costs led to the success of turbojet and turbofan engines, which are lower in propulsive efficiency but reach higher cruise speeds than propellers. With the world energy crisis in 1973-1974, interest in more efficient propeller systems arose, ultimately leading to research on unconventional concepts such as over-the-wing propellers, nowadays. However, while high-speed propeller concepts were first developed for monetary reasons as they promised an improved efficiency compared to turbofans, hence, less fuel consumption, an increased awareness of the impact of air travel on the climate, led to the revival of propeller-aircraft research today.

The research conducted for this thesis is part of a larger research program (Horizon 2020 Clean Sky 2 Large Passenger Aircraft program) funded by the European Union which ranges from investigating the aerodynamic interaction effects of over-the-wing propellers for distributed propulsion [40] to deriving a preliminary sizing method of a hybrid-electric passenger aircraft featuring over-the-wing distributed-propulsion [18]. The following sections, 1.1 and 1.2, will place the present research more on the fundamental side of studying the aerodynamic interaction effects within the larger research framework as it elaborates on the underlying research objectives and the outline of this thesis.
1.1. Research Objective

A problem that can arise when placing a propeller closely over a wing is flow separation over the flap, due to the interaction of the propeller with the wing, leading to a loss in lift [48]. This is particularly imminent at low advance ratios and high-thrust settings, e.g. at take-off, as the aerodynamic interaction effects are strongest at these conditions [40]. Thus, the high-thrust condition, particularly in combination with a deflected flap, is a crucial setting for the evaluation of an OTW system. A suggested solution to prevent flow separation on a deflected flap is tilting the propeller with the flap, to redirect the propeller's slipstream onto the flap, hence, providing a means of thrust vectoring and flow control, as well as exploiting the Coanda Effect [82]. Furthermore, the unsteady RANS simulations in Ref. [48] also indicate a lack of accuracy of steady-state simulations, which are conventionally used. Hence, not only the OTW system will be assessed but also the capability of the unsteady RANS simulations used in this thesis to capture the aerodynamic interaction effects accurately.

The objects to be investigated are the propeller and the wing with flap. There is a two-way interaction to be studied: the effect of the propeller on the lift and drag of the wing and the influence of the wing on the propeller thrust. Structural, stability, and noise analyses are not involved as they would exceed the scope of this project. However, they are important topics for future research. The nature of the proposed research project is practice-oriented as the outcome should be additional knowledge contributing to the successful design of distributed OTW propeller aircraft. From a fundamental point of view, it is necessary to identify the dominant flow phenomena on a wing with an OTW propeller. Therefore, the wing features a zero-pressure-gradient upper surface to isolate the influence of the deflected flap on the local pressure distribution. That being said, it needs to be assessed how accurately unsteady RANS simulations really can predict the interaction effects, not only in terms of global coefficients such as lift and drag, but also flow separation. Only then, the high-lift performance of a wing with an OTW propeller can be analysed. Conceptually speaking, this can be put into three stages: Diagnosis, Test, and Evaluation. The problem of flow separation has been identified in a literature study and needs to be diagnosed in the present study by identifying the dominant flow phenomena on the wing due to the interaction with the propeller. Then, the proposed solution of an inclined propeller will be tested. Finally, it needs to be evaluated whether the novel design indeed solves the problem of flow separation and what impact this has on the forces of the combined system. Hence:

The objective of the research project is to contribute to the development of distributed OTW propeller systems by studying the aerodynamic interaction between a propeller above a deflected flap and the wing, through unsteady RANS simulations.

Based on the research framework and objective, the following research questions were defined:

1. How well is the aerodynamic interaction between the OTW propeller and wing with adverse pressure gradient predicted by unsteady RANS simulations?
   (a) Which flow phenomena are captured?
   (b) What is the difference compared to wind tunnel experiments?

2. Which dominant aerodynamic interaction effects occur in a system comprising a propeller at high thrust above a deflected flap of a wing?
   (a) How is the wing pressure distribution affected by the propeller?
   (b) How does the flow develop over the flap?
   (c) How does the propeller interact with the boundary layer on the wing?
   (d) How is the aerodynamic interaction affected by wind tunnel walls?
   (e) How do the observed flow phenomena change when the propeller is inclined?

3. What is the impact of the aerodynamic interaction effects on the forces generated by the system?
   (a) What is the effect on propeller thrust and in-plane forces?
   (b) How are the wing lift and drag affected?
   (c) What is the impact on the overall propulsive efficiency of the system?
   (d) How are the forces affected when the propeller is inclined?
1.2. Thesis outline

The outline of this thesis is as follows. First, a literature study explaining the necessary theory and providing a critical overview of relevant research will be presented in Chapter 2. Then, the methodology of the present work will be explained in Chapter 3, starting with the experimental and numerical set-up, and followed by a verification and validation study of the isolated wing and propeller in Chapter 4 and Chapter 5, respectively.

Subsequently, a brief analysis of the aerodynamic characteristics of the individual components will be given in Chapter 6. This will be followed by a discussion of the results of the integrated, so-called baseline, configuration in Chapter 7. It is called baseline because, in this configuration, the propeller axis will not yet be inclined. The chapter about the baseline configuration will be structured such that the results will be compared to the wind tunnel experiments first. Then the influence of the wind tunnel walls on the aerodynamic interaction effects will be studied, followed by an analysis of the aerodynamic interaction effects without wind tunnel walls, and finally, the resulting forces on the propeller and wing will be presented.

After the study of the baseline configuration, the inclined configuration will be evaluated in Chapter 8, starting with an analysis of the aerodynamic interaction effects. Then, similar to the baseline configuration, the resulting propeller and wing forces will be explained. Finally, a qualitative performance comparison between the baseline and inclined configuration will be given, followed by the conclusions to the research questions in Chapter 9 and recommendations for future work in Chapter 10.
Theory and Literature Review

One of the first studied over-the-wing configurations was the Custer Channel Wing in the early 1950s [17, 53]. While this design has the highest potential for noise shielding [47], recent studies by Müller and the Collaborative Research Centre SFB 880 of the Technische Universität Braunschweig [45, 48, 49] showed that in terms of lift, drag and propeller efficiency a less integrated propeller over a plain wing represents an aerodynamic optimum. With the increased interest in distributed propulsion [36, 67, 78] enabled by improvements in electric or hybrid-electric, propulsion [61] a trend in research on reduced propeller diameter [40] and increased number of propulsors [56] can be identified. Propellers are not only a means of generating thrust but can also be used to actively increase lift, instead, either as a separate high-lift system [54, 55] or combining thrust generation and lift augmentation by thrust vectoring [12, 40].

In the following sections a review of research on OTW propeller systems will be presented. In order to understand the complex interaction effects between a wing and an OTW propeller, it is helpful to first familiarise oneself with the isolated propeller aerodynamics. This is the purpose of the following section 2.1. Subsequently, the aerodynamic interaction effects of the wing on the propeller performance, and of the propeller on the wing performance, will be discussed in section 2.2 and 2.3, respectively. Then, in section 2.4, considerations on the influence of an OTW propeller on the wing boundary-layer will be given. Finally, section 2.5 presents different computational methods available for flow simulations of OTW propeller systems.

2.1. Propeller aerodynamics

The rotating blade of a propeller is subjected to a relative flow velocity which partially consists of a component in the rotational direction, the tangential speed \( \Omega r \), and in the axial direction, the free-stream velocity \( V \). Additionally, velocities in tangential and axial direction are induced, due to the accelerated, swirling flow in the propeller’s slipstream. A sketch of the velocities at a blade element is shown in Figure 2.1. The component induced in the tangential direction \( v_{t0} \) stems from the rotation of the wake, which is necessary for the conservation of angular momentum when the propeller imparts torque on the flow. This induced velocity is directed oppositely to the rotational speed, whereas the induced velocity along the free-stream axis \( v_{x0} \) has the same direction as the free-stream velocity. The resulting velocity \( U \) forms an angle of attack \( \alpha \) with respect to the chord line of the airfoil.

The forces and moments, i.e. lift \( dL \), drag \( dD \) and torque \( dQ \), at a blade element can be derived from the velocities at the element’s radial position. The thrust \( dT \) of a blade element is the resulting force vector in axial direction. Integrating the elemental thrust and torque over the blade radius and multiplying with the number of blades leads then to the net thrust and torque of the propeller. This procedure, with its underlying mathematical models, is called Blade Element Theory.

Alternatively, the force of the propeller on the surrounding fluid, hence the thrust, can also be calculated with the integral description of the momentum conservation applied to an appropriately closed control surface \( S \):

\[
\int_S (p n + \rho n \cdot uu) dS - F = 0
\]  

(2.1)

Where \( n \) is the unit vector pointing outward of the control surface \( S \), \( u \) is the velocity vector at the surface and \( F \) is the force exerted on the fluid. The propeller is assumed to have an infinite number of blades resembling
2. Theory and Literature Review

Figure 2.1: Velocities at a blade element. Figure adapted from Ref. [75]

a disc over which the thrust is distributed uniformly. This is called an actuator disc. If the control surface is chosen to follow the contour of the slipstream, the boundaries are \( S'_0, S'_1 \) and \( S'_2 \), as shown in Figure 2.2. The momentum integral over the stream tube surface \( S'_1 \) is zero, since the velocity \( u \) and the normal vector \( n \) are perpendicular to each other \((n \cdot n = 0)\), and due to the symmetry in pressure. Therefore, the conservation of momentum from equation (2.1) simplifies to:

\[
F = \int_{S'_2} [(p_2 - p_0) + \rho u_2(u_2 - u_0)] dS_2
\]

Here, the primes have been dropped as the integrand is zero outside of the slipstream. Thus, two surfaces \( S_0 \) and \( S_2 \) of equal area can be used, as long as they completely enclose the streamtube’s cross-section.

Figure 2.2: Control surface for the momentum integral, from Ref. [75].

Inside the streamtube, the velocity increases gradually across the actuator disc, while total and static pressure jump at the disc location, as shown in Figure 2.3. The thrust produced is directly proportional to the axial velocity increase. Swirl in the propeller slipstream does not contribute to the axial velocity increase and, therefore, can be seen as wasted momentum [75]. Along the slipstream, the static pressure decreases in front of the disc. Then, it jumps at the disc location and decreases again to its initial value.

In practice, the Blade Element Theory and the Actuator Disc Theory are usually combined as Blade Element Momentum (BEM) Theory. While the conservation of momentum equation is restricted to the axial velocity increase, the Blade Element Theory also accounts for the angular momentum of the propeller. For clarity, the resulting thrust and torque are conventionally expressed as non-dimensional coefficients. Two commonly used definitions are, respectively:

\[
C_T = \frac{T}{\rho n^2 D_p^4}
\]

\[
C_Q = \frac{Q}{\rho n^2 D_p^5}
\]

(2.3)

(2.4)
Where $n$ is the propeller’s rotational speed, and $D_P$ is the propeller diameter. Another non-dimensional parameter is the advance ratio $J = \frac{V}{nD_P}$, which relates the free-stream velocity $V$ to the rotational speed. How the advance ration affects the thrust of a blade can be explained with Figure 2.1. A decrease in advance ratio means an increase in rotational speed, or decrease in free-stream velocity, or both. Thus, the angle of attack $\alpha$ between the resulting velocity $U$ and the wing is increased. As a result, more lift and, hence, more thrust, is produced by the blade. The propulsive efficiency of a propeller can be defined as $\eta_{pro} = \frac{T}{P_s}$, where $P_s$ is the shaft power.

![Figure 2.1: Axial velocity $v_x$ increase and both total $p_t$ and static $p_s$ pressure development across the actuator disc. The pressures are not to scale. Figure adapted from Ref. [72].](image)

The performance of a propeller interacting with a body can be analysed by considering the propeller as an actuator disc. In case the propeller is subjected to a locally increased or reduced flow velocity, the initial velocity through the disc $(V + v_{x0})$ is modified by a factor $m$. For simplicity, the flow here will be treated as a uniform potential flow without viscous effects. The slipstream far behind the propeller disc is unaffected by the local velocity disturbance and the net thrust on the propeller and body can be calculated with the conservation of momentum (Equation (2.1)) at this downstream location:

$$T_{\text{net}} = \rho(\nu + v_x) v_x S$$  \hspace{1cm} (2.5)

A characteristic property of the actuator disc is that the velocity induced at the disc is half of the maximum velocity induced in the slipstream: $v_{x0} = v_s/2$ [72]. By using this relation and the continuity equation for the slipstream cross-section, Wald [75] showed that the actual thrust at the disc is then:

$$T = \Delta p S_0 = \rho(V + v_x) v_s S/m$$  \hspace{1cm} (2.6)

Hence,

$$T_{\text{net}}/T = m$$  \hspace{1cm} (2.7)

For a propeller operating in a region of increased velocity, $m > 1$. Thus, the thrust on the system of body and propeller $T_{\text{net}}$ is larger than the thrust $T$ of the propeller itself. This increases the net propulsive efficiency of the system and can be attributed to the ingestion of a larger mass flow of air by the propeller. The converse
is true for a propeller operating in an area of reduced velocity. However, with increasing inflow velocity the blade angle of attack decreases, which reduces the thrust produced by the propeller [74]. Hence, a higher rotational speed component of the inflow velocity at the blade, i.e. higher rounds per minute of the propeller, is necessary to maintain the same propeller thrust in a high velocity flow. The relation by Wald might hold in a simplified, conceptual, environment but is ambiguous regarding how the propeller thrust is affected. This will be presented in the following section.

### 2.2. Aerodynamic effects of the wing on the propeller performance

The efficiency of an OTW propeller decreases, for a given geometry and constant shaft power, due to two flow characteristics inherent to an OTW design: the increased inflow velocities and the vertical velocity gradient above the wing. Both affect the effective blade angle of attack and therefore, the thrust produced by the blades. The velocity increase above the wing is not uniform but becomes stronger closer to the wing surface, for vertical locations outside the wing’s boundary layer. Hence, there is a vertical velocity gradient, which alters the effective blade angle of attack such that less thrust is generated from the part of the propeller disc closer to the wing. Even if the propeller would be designed for the general higher inflow velocities above the wing compared to the free stream velocity, segments of the propeller would still operate in off-design condition due to the vertical velocity gradient.

In early wind tunnel experiments, this adverse effect on the propeller efficiency was sometimes deemed negligible in take-off condition [16, 34] or not present at all, but actually an improved propeller performance was reported instead in cruise condition [34]. More recent RANS CFD simulations, however, showed a decrease in efficiency of 20% for a propeller placed above the mid-chord of a wing compared to a tractor propeller, due to the accelerated non-uniform inflow [47, 49]. Furthermore, it was concluded that the increase in inflow velocity has a larger impact on the aerodynamic propeller performance than the vertical velocity gradient. However, the non-uniformity of the inflow should not be neglected, as it leads to a cyclic propeller loading which may cause additional vibrations and interaction noise [47]. The resulting propeller disc loading is shown in Figure 2.4, comparing an isolated propeller, a tractor propeller, an OTW propeller and a propeller embedded in a channel wing. From the non-uniform distributions of thrust generated over the over-wing disc area (Figure 2.4c), it can be inferred that little to no thrust is produced close to the wing. Comparing the propeller disc loading of the channel wing (Figure 2.4d) to the over-wing configuration suggest that, a stronger integration of propeller and wing leads to a stronger adverse effect on the propeller performance. This is confirmed in Ref. [45], where it is shown that the thrust of a propeller integrated in a channel wing with the depth of half the propeller diameter, is 11% lower compared to a plain over-wing propeller.

While for the OTW propeller analysed in Ref. [45] \( (D_p/c = 1.32) \) no significant impact of the axial position on the propeller performance was found, other authors [40] found an improvement in efficiency of a smaller propeller \( (D_p/c = 0.395) \) for an axial position close to the wing’s trailing edge. The flow over a wing decelerates toward the trailing edge after it passed the chordwise position of maximum velocity, given there is no velocity increase again due to a flap. Hence, the decrease in thrust due to the increased inflow velocities is smallest at the trailing edge. Apparently, this interaction is more pronounced for smaller propellers, as they are more affected by the local supervelocities above the wing. Further improvements in propeller performance could be possible by inclining the propeller according to the curvature of the wing's upper surface or even deflecting it with a flap. This improves the axial alignment of the propeller with the streamlines above the wing and has shown to increase the momentum in the slipstream [40]. However, research on inclined OTW propellers is limited and even the one on inclined tractor configurations is sometimes contradictory or highly dependent on the flight condition and coefficient definitions [25].

For isolated propellers it is known that, when the propeller is positioned at a positive angle of attack to the incoming flow (thrust axis pointing more upward), the up-going blade experiences a decrease in angle of attack whereas the down-going blade experiences an increase in angle of attack [74]. Furthermore, the angle of attack increases more for the down-going blade than it decreases for the up-going blade which leads to a slightly higher propeller thrust. However, since the thrust axis is not aligned with the longitudinal aircraft axis, the thrust in the flight direction is smaller. On the other hand, for a propeller installed in front of a wing it can be beneficial to tilt the propeller downward to reduce the impact of the flow non-uniformity of the wing induced upwash on the propeller. In such a configuration also the lift of the wing is increased due to the increased upflow in front of the wing induced by the propeller. The resulting force vector acting on the wing is tilted more forward which reduces the induced drag and benefits the installed thrust [71].
2.3. Aerodynamic effects of the propeller on the wing performance

Figure 2.4: Distributions of effective blade angle of attack and relative local thrust at $\alpha = 0$, from Ref. [46].

The effect of OTW propeller inclination on the wing might be completely different. However, similar reasoning of aligning the propeller with the local flow field would lead to a tilt back (thrust axis pointing more upward) of the propeller due to the curvature of the airfoil, if placed behind the location of maximum airfoil thickness. The research in Ref. [40] shows that this can indeed improve the propeller performance due a better alignment with the streamlines. As a result, also the pressure distribution on the wing in front of, and behind the propeller is changed. The impact of a tiltable OTW propeller on the wing aerodynamics will be further explained in the next section.

2.3. Aerodynamic effects of the propeller on the wing performance

An OTW propeller can locally increase $C_L$ by changing the pressure distribution, thus altering the effective airfoil shape [56]. The highest lift augmentation can be reached with a propeller at the trailing edge, as close as possible to the wing surface, as this extends the suction area on the wing and increases suction in front of the propeller, respectively, the most [16, 34, 45, 47].

A propeller represents a jump in pressure, as seen in Section 2.1. Hence, the flow behind an OTW propeller is more prone to separate due to the adverse pressure gradient at the propeller location. Over a flap, flow separation can be prevented by an internally blown flap (IBF) as it increases the momentum in the boundary layer [38, 47]. When applied to the propeller’s slipstream, a stronger flow deflection due to later separation of the flow over the flap, can be exploited for high lift through an increase in circulation [2, 24]. However, this kind of indirect thrust vectoring is better suited for a tractor propeller, since for an OTW propeller the contracting slipstream diverges from the wing surface which may cause flow separation [47]. Flow separation over the flap would reduce the lift, and increase the pressure drag of the wing. Alternatively, the propeller could be placed at an angle of attack to redirect the slipstream onto the flap, thereby increasing the momentum in the boundary-layer, and actively augment the lift through propeller thrust vectoring as done in Ref. [40]. The results in Ref. [40] show that inclining the propeller (configuration 5 in Figures 2.5 & 2.6) as well as deflecting the propeller together with the flap (configuration 6 in Figures 2.5 & 2.6), can lead to a larger lift increase compared to a configuration without propeller deflection (configurations 3 & 4 in Figures 2.5.
The pressure drag for both inclined propeller configurations, is even negative, and smaller than for the vertical propeller configurations, over all advance ratios tested. Hence, the increased lift and decreased pressure drag with an inclined propeller configuration leads to the hypothesis that such an OTW propeller configuration can prevent flow separation over the flap. This assumption will be tested by CFD simulations in this thesis.

Typically, the drag of a wing with OTW propeller is reduced due to the higher velocity in the intake streamtube in front of the propeller, inducing thrust at the leading edge [34]. In general, there is consent that an OTW propeller decreases the pressure drag [34, 40, 47]. Furthermore, it is agreed that the chordwise propeller position for minimum drag is ahead of the one for maximum lift, however, the locations given vary between 20% [34] and 60% [16] chord. Factors that could affect the propeller location for minimum drag are the wing’s angle of attack, flap deflection, and airfoil shape. While the angle of attack and flap deflection in Ref. [16] did not affect the axial propeller location for maximum lift increase, they did affect the location for minimum drag. Maximum drag reduction was achieved at a more forward location for increasing angle of attack of the wing, and more aftward for increasing flap deflection. Other authors found a maximum in drag reduction, at zero angle of attack, for a propeller positioned at the airfoil’s location of maximum thickness [40] or at the midchord [72]. Therefore, it is reasonable to assume that discrepancies in the axial propeller position for minimum drag in literature are also due to the differences in airfoils used.

Nevertheless, the chordwise position of the propeller for maximum drag reduction is further upstream than the location for maximum lift increase, as this increases the induced thrust at the leading edge. Hence, a trade-off needs to be made on the design for an acceptable lift-to-drag ratio. Preliminary design studies show that, for a reference aircraft with an OTW propeller mounted above the midchord of the wing, a corrected lift-to-drag ratio and the system’s propulsive efficiency can be increased by 27% and 34%, respectively, compared to a tractor configuration [47]. The corrected lift-to-drag ratio is defined as:

$$\frac{L}{D^*} = \frac{C_L}{C_D^*} = \frac{C_{L,\text{ref}} + \Delta C_L}{C_{D,\text{ref}} + \Delta C_D^*}$$  \hspace{1cm} (2.8)$$

The lift and drag coefficients $C_{L,\text{ref}}$ and $C_{D,\text{ref}}$ are for the reference aircraft at take-off. The increments $\Delta C_L$ and $\Delta C_D^*$ are due to the change lift and drag, when the propeller is installed. It is called the corrected lift-to-drag ratio, since the drag coefficient increment is corrected by the change in thrust coefficient $\Delta T_c = T_c - T_{c,\text{iso}}$ ($T_{c,\text{iso}}$ is the isolated propeller thrust coefficient) as $\Delta C_D^* = \Delta C_D - \Delta T_c$. The propulsive efficiency of the system is then defined as:

$$\eta_{\text{pro}} = \frac{T_{\text{inst}} V}{P_s}$$  \hspace{1cm} (2.9)$$

Where $T_{\text{inst}} = T - \Delta D$ is the installed thrust.

Another design parameter affecting the lift and drag of a wing with OTW propellers, is the size of the gap between the propeller tips and the wing surface. While the lift of the wing only increases with decreasing tip clearance [45], the drag exhibits a more complex behaviour. Often it is stated in literature, that the propeller should be as close as possible to the wing as this is supposed to reduce drag the most [16, 34, 40]. However,
2.3. Aerodynamic effects of the propeller on the wing performance

Figure 2.6: Wing lift and pressure drag increase with respect to the propeller-off conditions in high-lift configuration, from Ref. [40].

CFD simulations [45] reveal an increase in drag for gap-to-diameter ratios below 0.02. An explanation for this behaviour could be that the flow separates at the propeller location due to the too small tip clearance causing increased pressure drag. Literature on the interaction between an OTW propeller and a wing boundary layer is limited, however, unsteady RANS simulations of a channel wing [48] show that the unsteady effects of the tip vortices and pressure fluctuations can lead to a weakening of the boundary layer and eventually flow separation behind the propeller. Even the subsequent IBF in Ref. [48] was not strong enough to prevent the flow from separating as shown in Figure 2.7.

Figure 2.7: Steady (left) and unsteady (right) simulation of the flow field around a channel wing in a vertical plane through the propeller axis, from Ref. [48].

As a consequence, the suction peak at the flap is reduced (Figure 2.8), which reduced lift but also drag. The resulting lift-to-drag ratio in the unsteady simulations is almost 8% higher than the one obtained in steady simulations. Hence, relying solely on steady RANS simulations for the analysis in high lift configuration could be insufficient. However, there is the possibility that flow separation due to the unsteady interaction of the tip vortices with the boundary layer only occurs for channel wings because of their larger spanwise extent of minimal propeller clearance than a plain wing and only at high thrust settings or low advance ratios. No simulations were done for a propeller over a plain wing. In this context, it is essential for a distributed propulsion application of OTW propellers to investigate whether there is flow separation behind the propeller as this would render a high lift flap ineffective. With this in mind, the following section will
elaborate on which design parameters might affect flow separation, based on experiments of a propeller in close proximity to a flat plate.

![Figure 2.8: Unsteady pressure distributions of the channel wing below the propeller axis, from Ref. [48].](image)

2.4. Propeller–boundary-layer interaction
The interaction of a propeller and a wall in close proximity was first studied for maritime applications to understand the so called Propeller-Hull Vortex Cavitation (PHVC); a phenomenon where a vortex with cavitating cores extends from the ship hull to the propeller blades [33]. Sato et al. [60] showed experimentally in a water tunnel, with a propeller over a flat plate, that flow reversal on the plate only depends on the ratio of blade tip clearance to propeller diameter \( \frac{\epsilon}{D_p} \), and the relation between thrust and advance ratio \( T_c = 8C_T l/(\pi J^2) \). The flow was more likely to reverse for a small tip clearance and high propeller loading starting at a tip clearance of 0.3 \( D_p \) for a thrust coefficient of \( T_c = 8.65 \). According to the results of Sato et al. [60] flow reversal occurs when:

\[
T_c > \frac{16}{0.383} \left( \frac{\epsilon}{D_p} + 0.124 \right)^2
\] (2.10)

These experiments already highlight important design parameters for OTW propellers, however, it should be kept in mind that the working fluid was water over a flat plate. For a propeller over a wing working on air there might be additional parameters such as the boundary layer thickness or the adverse pressure gradient. Noise measurements of a similar configuration in air, however, were concluded to be consistent with the observations on marine propellers [30]. Furthermore, the results of Sato et al. were successfully replicated by an unsteady RANS solver [42].

Wind tunnel experiments by Wisda et al. [83], motivated by the acoustic analysis of a rotor partially immersed in a planar turbulent boundary layer, revealed a strong adverse pressure gradient immediately ahead of the rotor in a high thrust setting, at \( J = 0.58 \). The wind tunnel experiments were conducted at a wind speed of 20 m/s where the boundary layer had a thickness of 100 mm (0.22 \( D_p \)) at the location of the propeller. The gap between the propeller blades and the wall was 20 mm (0.044 \( D_p \)). Consequently, the propeller cut through 80% of the boundary layer thickness. When a rotor blade penetrates the boundary layer, the blade angle of attack at the tips is changed due to the reduced axial velocity inside the boundary layer. However, results in Ref. [83] suggest that this influence of the boundary layer on the propeller performance is minimised.
at low advance ratio since the rotational component at the blade tips dominates. The lowest pressures on the wall could be found 20% to 30% blade radius in front of the propeller. Between this low pressure area and the rotor plane the adverse pressure gradient formed due to air being sucked away from the wall into the rotor. This caused flow separation, which was also identified by Alexander et al. [29]. In Ref. [83], the instantaneous PIV flow field 10 mm away from the wall showed highly unsteady features. Two contra-rotating vortices at the propeller position were identified (Figure 2.9), indicating an unsteady arch vortex, which was also observed in the results of RANS simulations by Glegg et al. [30]. It is speculated that this arch vortex is responsible for flow separation as well [83]. The CFD simulations were done for the same propeller-wall set up like the one from Wisda et al. and showed, depending on the advance ratio, that flow reversal either occurred in front of the rotor \((J = 0.48)\) or behind the rotor \((J = 0.72)\). For an advance ratio greater than \(J = 0.8\) no flow reversal was identified.

More detailed wind tunnel experiments by Murray et al. [51] of the same propeller-wall set-up showed not only a strong low pressure area in front of the propeller but also a small secondary low pressure area directly below rotor disc for high thrust. This secondary low pressure was due to flow reversal. As shown in Figure 2.10, for high thrust, reversed flow was sucked from behind the propeller into the tip gap which produced the secondary low pressure field. (The colourbar-label in Figure 2.10 suggests only positive values, hence no flow reversal. However, this is probably a mistake by the authors and what is really shown is the magnitude of the mean velocity \(|U|\), as in Figure 2.9.) In these experiments, the wall constrained the mass flow of air in front of the disc, hence air was sucked through the propeller gap as compensation. The previously mentioned unsteady vortex structures at the propeller location were identified again. In conclusion, two mechanism can cause flow separation: reversed flow ahead and behind the propeller due to a wall limiting the slipstream contraction, and an unsteady vortex system in the tip gap. It is unclear to what extend this vortex system is influenced by the blade tip vortices. However, its resemblance to the so-called ground vortex found for aero engines [50] suggests that it is actually a result of the reversed flow from behind the propeller. This implies that the wall, representing a flow boundary to the streamtube, is the dominant influence on flow separation on the wall at the propeller location.

The gap to diameter ratio used by Wisda et al. [83], Glegg et al. [30] and Murray et al. [51] of about \(\epsilon/D_p = 0.044\) is larger than the one of Müller et al. [48] \(\epsilon/D_p = 0.02\) and Marcus et al. [40] \(\epsilon/D_p = 0.025\). This supports the findings of Müller et al., indicating flow separation, and suggest, the possibility of flow separation.
separation for Marcus et al. as well. However, in both papers a boundary layer thickness is not specified. It is therefore not clear if the propeller tips also penetrated the boundary layer. A thrust coefficient was not measured for the flat plate experiments. An application of the relation of Sato (2.10) to the test cases of Müller et al. ($\epsilon/D_p = 0.02, C_T = 0.32, J = 0.72$) and Marcus et al. ($\epsilon/D_p = 0.025, C_T = 0.12, J = 0.7$) suggests flow reversal and no flow reversal, respectively. However, in Ref. [40] only the isolated propeller thrust coefficient is available.

![Figure 2.10: Mean velocity contours and vectors through the tip gap for moderate thrust $J = 1.05$ (top) and high thrust $J = 0.58$ (bottom), from Ref. [51]. The black lines project the rotor on the wall in the z-x-plane.](image)

### 2.5. Computational methods for propeller interaction studies

There is a two-way interaction between a propeller and a wing in close proximity. The propeller influences the pressure distribution on the wing, and the wing alters the inflow to the propeller. Neglecting the influence of the wing on the propeller performance has often been done but was shown to be inaccurate when assessing the propulsive efficiency of the entire propeller-wing system [71].

For steady state simulations investigating only major integration effects, namely lift-to-drag ratio and propeller efficiency, it is sufficiently accurate to model the propeller as actuator disc [48, 73]. For unsteady simulations, however, a model of the full propeller geometry or at least an actuator line model is necessary to simulate three-dimensional unsteady effects such as the blade tip vortex and boundary layer separation [41, 48]. In an actuator line model momentum sources and energy sources distributed along lines represent the propeller blades. Accurate agreement in the time-averaged and time-accurate pressure distributions on a wing between a full-blade model and an actuator line model has been shown [65].
For inviscid calculations, the wing geometry can be modelled with a panel method [16, 40], based on the claim that the effect of an OTW propeller on the viscous drag of an airfoil is negligible [34]. Furthermore, a Vortex Lattice Method (VLM) can be used to quickly analyse several conceptual design configurations [40, 68, 71]. This might show acceptable agreement with experiments in cruise condition [37, 40]. For large flap deflections [40] and high thrust coefficients [37] (i.e. if there is flow separation), however, there is poor agreement between numerical and experimental results. In case there is no flow separation, the lift coefficient can be obtained accurately with inviscid methods. Drag predictions, however, are limited due to difficulties in estimating the profile drag [71]. Due to these limitations in drag prediction and flow separation as well as the interaction of the propeller tip vortices with the wing boundary layer it is clear that viscous effects need to be taken into account when assessing the high-lift performance of a wing with OTW propellers. One of the most common CFD techniques for simulating propeller-wing interactions, capable of capturing viscous effects, is performing Reynolds-Averaged Navier-Stokes (RANS) calculations.

As the name suggests, Reynolds-Averaged Navier-Stokes simulations give averaged solutions to the Navier-Stokes equations. All the turbulence in the flow is modelled by empirical relations instead of being resolved. Steady state as well as unsteady solutions can be obtained. The underlying idea of the RANS equations is to derive simpler equations from the fundamental conservation laws for turbulent flows. In order to do so, the flow variables are composed in their mean value and fluctuation. For instance, for the velocity \( u_i \) this would result in \( u_i = \bar{u}_i + u'_i \) where \( \bar{u}_i \) is the averaged value and \( u'_i \) the fluctuation. Substituting this sum into the Navier-Stokes (NS) equations and applying a Reynolds averaging process leads to the Reynolds Averaged Navier-Stokes equations. When averaging non-linear terms in the NS equations, additional terms arise, the so-called Reynolds stresses \( -\rho u'_i u'_j \). These lead to a closure problem, since there are more variables than equations. The only way to close the equations is to use empirical approximations, i.e. turbulence models. The most common turbulence models can be split into two groups: the Eddy Viscosity Models (EVM) and the Reynolds Stress Models (RSM).

It is known that turbulence leads to momentum exchange between fluid elements. Furthermore, it can be observed that this momentum exchange is facilitated by the viscosity [26]. This led to the eddy-viscosity hypothesis (2.11), which assumes that the effect of turbulence can be modelled as an increased viscosity (Eddy Viscosity Models):

\[
- \rho u'_i u'_j = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \rho \delta_{ij} k
\]  

(2.11)

where \( \mu_t \) is the increase in viscosity, the eddy-viscosity, \( \rho \) is the density, \( \delta_{ij} \) is the Kronecker delta and \( k \) is the turbulence kinetic energy \( k = \frac{1}{2} u'_i u'_j \). The term \( \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \) represents the shear rate. It should be noted that the eddy-viscosity \( \mu_t \) is a field quantity and not a material quantity as the dynamic viscosity \( \mu \). How the eddy-viscosity is expressed depends on the turbulence model used. The models usually are characterised by the number of required transport equations. There are algebraic 0-equation models [3, 58] and models with one or more transport equations [35, 63, 79]. Each model represents a compromise between physical modelling and computational cost. In general, the more sophisticated the model is (more transport equations), the higher the accuracy of the model is compared to experimental results, but also the higher the computational cost (Table 2.1). A well-known one-equation model for aerospace applications is the Spalart-Allmaras model [63]. Two-equation models usually give an expression not only for the eddy-viscosity \( \mu_t \) but also for a turbulence length scale \( l_t \). The most frequent two-equation models relate these quantities to the turbulence kinetic energy \( k \) and the turbulence dissipation rate \( \epsilon \), or the turbulence specific dissipation \( \omega \). Hence, they are also called \( k-\epsilon \) or \( k-\omega \) models. Prominent examples are the Jones-Launder \( k-\epsilon \) model [35] and the Wilcox \( k-\omega \) model [79]. The model of Jones and Launder is based on an assumption of isotropic turbulence which does not hold in boundary layers and most real flows [80]. Therefore, it should be applied only to flows without strong pressure gradients, stream line curvature or separation. Wilcox's turbulence model gives good results for boundary layer flows and for flows with pressure gradients and separation but is strongly influenced by the free stream conditions outside the boundary layer [43].

New turbulence models are still developed and older ones are being improved or combined. A blending of the Wilcox \( k-\omega \) model for the inner region of the boundary layer with a standard \( k-\epsilon \) model for the outer region and the free shear flow resulted in the Shear Stress Transport (SST) \( k-\omega \) model by Menter [44]. This model seems superior to other one- or two-equation models, especially for high lift flows as well as predicting stall characteristics [11], and has already been used successfully to study the flow around a wing with distributed propellers at the leading edge [66, 67]. For investigations with an OTW propeller in high
Table 2.1: Estimated CPU time of various two-equation models for a fixed number of iterations relative to the CPU time of the Spalart-Allmaras model, from Ref [11].

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>CPU time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spalart-Allmaras</td>
<td>1.00</td>
</tr>
<tr>
<td>Wilcox $k - \omega$</td>
<td>1.25</td>
</tr>
<tr>
<td>TNT $k - \omega$</td>
<td>1.28</td>
</tr>
<tr>
<td>Myong-Kasagi $k - \epsilon$</td>
<td>1.33</td>
</tr>
<tr>
<td>Menter SST $k - \omega$</td>
<td>1.39</td>
</tr>
<tr>
<td>$k - \epsilon$ MK+Shih (2)</td>
<td>1.45</td>
</tr>
</tbody>
</table>

lift configuration, accuracy in predicting flow separation is of great importance and, therefore, a turbulence model should be chosen that actually resolves the flow in the boundary layer as well as possible.

There are limitations to the eddy-viscosity models due to the assumption that the Reynolds stresses are proportional to the mean shear rate. They cannot distinguish effects of the individual components of the Reynolds stress tensor. For instance, the two-equation models are said to over-predict lift if a jet is blown over a wing, which was explicitly shown for the $k - \epsilon$ model [39]. Moreover, rather simple relations between the Reynolds stress and the strain rate which are correct in two dimensions may not correctly represent the more complex behaviour in three dimensions [26]. To overcome these deficiencies Reynolds Stress Models (RSM) can be used to directly solve model transport equations for all components of the unknown Reynolds stress tensor. Reynolds stress models give by design superior results for flows with strong streamline curvature or swirl, and for anisotropic turbulence induced secondary flow structures [62]. However, they are computationally expensive as they require at least 6, usually 7, transport equations. Furthermore, convergence can be slow or even unstable due to their stiffness [80]. However, there are so-called Explicit Algebraic Reynolds Stress Models (EARSM) which reconstruct the Reynolds Stress tensor or an anisotropic eddy viscosity tensor from a reduced number of transport equations. Such an $\omega$-based EARSM of Wallin and Johansson has been shown to be robust and give good results at low computational cost [76].

Higher fidelity simulation methods such as Large Eddy Simulations (LES) or Direct Numerical Simulations (DNS) exist, but come at a price of higher computational cost than RANS simulations. LES are very good in resolving flows with large turbulent structures. For boundary layer flows, however, the grid needs to be very fine close to the wall, which increases the computational cost significantly [77]. The inaccuracy in the representation of separated flows of RANS simulations can be overcome with LES. However, at high Reynolds numbers LES becomes, again, exceedingly expensive [7]. As an estimation, halving the grid spacing increases the necessary memory and CPU time by a factor of 8 and 16, respectively [57]. To reduce the computational cost while maintaining LES like accuracy, hybrid methods combining RANS and LES are developed; so called Detached Eddy Simulations (DES) [20]. DES switch from RANS simulations close to a wall to LES depending on the mesh definition. Applied to a flat plate at an incidence angle DES was able to predict unsteady vortex shedding, conversely to RANS simulations [9]. To simulate the flow around a high-lift configuration with a slat and flap is still very difficult due to the complex boundary layer flow [20]. DES are very sensitive to the mesh size. An incorrect grid spacing or definition of the cut between RANS and LES can erroneously cause flow separation and lead to completely wrong results [10]. That being said, simulations of an isolated marine propeller [52] comparing DES with RANS simulations showed that integral quantities such as thrust and torque can be obtained equally well with a RANS solver. The DES results were more detailed; however, the increased computational cost was not justified by the improvement of the results in terms of propeller efficiency.
Methodology
This chapter describes the experimental and numerical set-up of the thesis. Since the geometry and configuration of the CFD simulations were chosen based on experimental data available for validation (see Ref. [19]), the set-up of these wind tunnel experiments will be described first. Then, the numerical set up will be explained, including the governing equations, turbulence modelling, an explanation of the overset mesh generation, as well as an overview of the computational domain and the boundary conditions used.

3.1. Experimental set-up

In this section, the test facility and the measurement techniques will be presented. The tools used for the measurement techniques are wing-surface pressure taps, an external balance, a pitot probe, and a Particle Image Velocimetry set-up.

3.1.1. Wind Tunnel Facility and Model

The experiments were conducted in the closed-circuit, low-speed, low-turbulence tunnel (LTT) of TU Delft. At the freestream velocity of $V = 20 \pm 0.05$ m/s used for this study, the inflow turbulence level was below 0.05%. The maximum attainable freestream velocity of the LTT is 120 m/s. The octagonal test section had a cross-section of $1.8 \times 1.25$ m, where the four diagonal corners had a length of $\sqrt{0.3^2 + 0.3^2}$ m. Inside the test section, the untapered wing model was placed vertically, at zero angle of attack, spanning the entire tunnel height as shown in Figure 3.1. The propeller could be placed with a horizontal support sting in close proximity to the suction side of the wing. This support sting could then be connected to a balance for force measurements or to a three-axis traversing system.

The wing model featured a plain flap which could be deflected 20 degrees. At 20 degrees deflection and $V = 20$ m/s, the flow around the isolated wing stayed attached over the flap hinge but started to separate towards

![Figure 3.1: Rear view of the wing and the propeller inside the test section including annotations for the individual components. Figure adapted from Ref. [19].](image-url)
the trailing edge, as was observed in preliminary test runs. For the deflected configuration, the chord length was 1.042 m due to a fowler motion of the flap, instead of the 1.02 m chord in the retracted state. A sketch of the side view of the wing profile and propeller, including their main dimensions, is given in Figure 3.2.

The wing was designed to have a zero pressure gradient in front of the propeller, such that a wing extension covering the flap, could be used to simulate the flow over a quasi flat plate and test its aerodynamic interaction with the propeller (not used in this thesis). Therefore, the wing profile consisted in front of the flap of two flat surfaces for the top and bottom sides, extending up to the flap hinge at 80% chord, with a thickness-to-chord ratio of 11%. The flap curvature represented a circular arc, tangent to the suction side, with a radius of curvature equal to approximately 10% of the chord. The front of the wing profile was modelled as a modified super-ellipse of aspect ratio 6 up to 32% chord. To enforce boundary layer transition, the boundary layer of the wing was tripped with a 3 mm wide strip of carborundum particles with an average diameter of 530 µm (grit size 36) located at 7.5% chord on both sides of the wing. The chord based Reynolds number of the wing model was $Re = 1.4 \times 10^6$.

![Figure 3.2: A sketch, including main dimensions, of the wing profile with the propeller above the flap hinge (left) and a close-up of the blade tip penetrating the wing's boundary layer. The deflection of the flap sketched here is larger than 20 degrees. Figure adapted from Ref. [19].](image)

The propeller was positioned above the flap hinge at $x/c = 0.79$ of the mid-span profile. This location was chosen based on previous studies which showed that mounting the propeller close to the trailing edge represents a good compromise between lift gain, drag reduction, and propulsive efficiency loss [40]. The propeller had six blades with a diameter of 0.2032 m ($D_p/c = 0.2$) and a blade pitch of $45\degree \pm 0.05\degree$ at 70% of the blade radius. The gap $\epsilon$, between the propeller tips and the wing surface, was 7.5 mm ($\epsilon/R = 0.0738$). This corresponds to 50% of the local boundary layer thickness $\delta_{99}$, when the suction side was extended as flat plate instead of the deflected flap. With $V = 20$ m/s, the propeller operated at an advance ratio of $J = 1.13$. The accuracy of the rotational speed of the propeller was in the order of $\pm 0.1$ Hz. Combined with the uncertainty in freestream velocity, this results in deviations from the desired advance ratio below $\pm 0.5\%$. The isolated propeller thrust at these operating conditions is $C_T = 0.35$. More details on the isolated propeller geometry and performance can be found in Appendix A.

### 3.1.2. Measurement Techniques

The goal of the experiments was to obtain the pressure distribution of the wing profile (with and without propeller), analyse the propeller performance, and visualise the flow over the flap. Therefore, pressure taps on the wing surface were used to measure the pressure distribution, an external balance was connected to the support sting of the propeller to measure the propeller thrust, and PIV measurements were taken to visualise the flow over the flap. Additionally, a pitot probe was used to determine the boundary layer thickness of the wing without the propeller present, at $x/c = 0.79$. The following paragraphs provide an overview of these measurement techniques.

**Wing surface pressure taps:** In total, 81 static pressure taps for time-averaged measurements were flush mounted in the wing surface. Towards the leading edge and trailing edge, more pressure taps were installed for an increased spatial resolution. The pressure ports were arranged in a zig-zag line with an offset of $\pm 15$ mm from the mid-span profile, as shown in Figure 3.3. This alternation in spanwise coordinate was chosen to
reduce disturbances from the respective previous hole, but could not be realised over the flap. Each pressure measurement was taken over 30 seconds, with an accuracy of ± 1 Pa due to the uncertainty of the measurement sensors. The post-processing of the pressure measurements was done in Matlab. A script was written to read the output file of the data acquisition and plot the pressures \( p \) of each pressure port as pressure coefficient \( C_p = (p - p_{\text{s,\infty}})/q_{\infty} \) against the axial coordinate of the wing, where \( p_{\text{s,\infty}} \) and \( q_{\infty} \) are the free-stream static and dynamic pressure, respectively, measured inside the test-section.

**External balance:** A six-component balance was used for measuring the propeller thrust. Axial forces up to 50 N could be measured with an uncertainty of ± 0.002 N. The acquisition time was 30 seconds, which corresponded to 2610 propeller revolutions, at \( V = 20 \text{ m/s} \) and \( J = 1.13 \). A dummy spinner without blades was used for propeller-off measurements to obtain the net forces generated by the propeller blades. The thrust coefficient was then calculated from the net thrust of the blades, meaning: \( T = T^{\text{on}} - T^{\text{off}} \). However, this thrust value still includes contributions of the change in drag of the nacelle, and support sting, due to the slipstream washing over it.

**Pitot probe:** The pitot probe was traversed in the positive \( Z \)-direction on the suction side of the wing, at \( x/c = 0.79 \), to measure the static and total pressure at the position of the propeller. The closest measurement to the wing was at a distance of 0.4 mm (0.0039 \( R \)). From there, the pitot probe traversed in steps of 0.25 mm to a distance of 5.4 mm, then, in steps of 0.5 mm up to a distance of 10.4 mm, and finally, in steps of 1 mm up to a distance of 80.4 mm (0.79 \( R \)). For each measurement point, the pressures were averaged over a period of 30 seconds. The boundary-layer profile and thickness was then obtained from the total pressure measurements, by using the total pressure coefficient \( C_{p,t} \):

\[
C_{p,t} = \frac{p_t - p_{t,\infty}}{q_{\infty}} + 1
\]

where \( p_{t,\infty} \) is the freestream total pressure, measured upstream of the wing model. The boundary-layer thickness, \( \delta_{99} \), was defined as the distance where \( C_{p,t} = 0.98 \), that is 0.99\( ^2 \) times the freestream total pressure. The total pressure sensors had an uncertainty of ± 4 Pa.

**Particle Image Velocimetry:** Stereoscopic PIV measurements were taken in a plane perpendicular to the wing, between the nacelle and the flap. The field of view (FOV) is shown in Figure 3.4. Flow seeding was achieved by a SAFEX Twin Fog DP generator using SAFEX Inside Nebelfluid, which generated particles with an average diameter of 1 \( \mu \)m and a relaxation time of 1 \( \mu \)s. The particles were illuminated by a 200 mJ Quantel Evergreen laser, directed through the designated slits in the wing. The resulting laser sheet had a thickness of 2 mm. Image acquisition was done at 15 Hz with two LaVision Imager sCMOS 16-bit cameras featuring a 2560 \( \times \) 2160 pixel sensor, a pixel size of 6.5 \( \mu \)m, and a Nikkor 105 mm f/8 lens. For time averaged results, 600 uncorrelated images were taken. To obtain phase-locked results, the cameras were synchronised with an external signal which triggered the image acquisition at a defined phase angle of the propeller. In this manner, 300 phase-locked images were taken. The images were then post-processed with the LaVision Davis.
8.4 software. First a time filter was subtracted, then a stereo cross-correlation was applied with \( 96 \times 96 \), followed by \( 24 \times 24 \), multi-pass iterations of 50% overlap, each. The main parameters of the PIV set-up are summarised in Table 3.1. For the comparison with the numerical results, the numerical flow data will be extracted from the same geometric field of view and both the experimental and numerical data are plotted as a Matlab contour plot.

Table 3.1: PIV set-up from Ref. [19]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Focal length [mm]</td>
<td>105</td>
</tr>
<tr>
<td>Field of view [mm(^2)]</td>
<td>100 \times 150</td>
</tr>
<tr>
<td>Pulse delay [(\mu)s]</td>
<td>25</td>
</tr>
<tr>
<td>Imaging resolution [pixel/mm]</td>
<td>19</td>
</tr>
<tr>
<td>Window size [pixel(^2)]</td>
<td>24</td>
</tr>
<tr>
<td>Overlap factor [%]</td>
<td>50</td>
</tr>
<tr>
<td>Vector spacing [mm]</td>
<td>0.6</td>
</tr>
<tr>
<td>Velocity uncertainty [%]</td>
<td>2.5</td>
</tr>
</tbody>
</table>

3.2. Numerical set-up

In the following sections, the numerical set up will be explained, starting with a recap of the governing equations to be solved by the commercial, cell-centered, finite volume ANSYS Fluent 19.1 solver (Section 3.2.1). Then, the turbulence model used in this study will be discussed in Section 3.2.2. Furthermore, an advanced technique of grid generation and solution calculation, called Overset or Chimera methodology, will be used in this thesis. Therefore, section 3.2.3 elaborates the mesh generation and calculation procedure of such an overset approach. Finally, Section 3.2.4, presents the computational domain with its boundary conditions.

3.2.1. Governing equations

As pointed out in section 2.5, RANS or higher-fidelity simulations are necessary for a wing with a deflected flap featuring an OTW propeller. The reasoning for viscous RANS calculations, instead of inviscid alternatives, is that literature shows a likelihood of flow separation on the wing, due to the presence of the OTW propeller. In case of flow separation, inviscid calculations agree poorly with experiments. Furthermore, a possible interaction between the blade tip vortices and the wing boundary-layer, influencing the wing's aerodynamic performance was identified. In order to study the behaviour of the boundary-layer of the wing or a propeller
blade, viscous simulations are inevitable. As part of the literature study it was also concluded that high-
fi delity simulations, such as LES, would likely increase the accuracy and details of the solution. However, this
improvement was not justified by the inherent increase in computational cost. Therefore, RANS simulations
will be used in this thesis.

RANS simulations solve a time-averaged form of the Navier-Stokes equations, which in case of an incom-
pressible flow are a set of equations comprising the continuity equation (conservation of mass (3.2)) and
Newton’s second law (conservation of momentum (3.3)):

\[
\frac{\partial u_i}{\partial x_i} = 0 \\
\rho \frac{\partial u_i}{\partial t} + \rho \frac{\partial}{\partial x_j}(u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(2\mu s_{ij})
\]

Equations (3.2) and (3.3) are described for a 2D flow, where \( u_{i,j} \) are the velocity components, \( x_{i,j} \) are the
spatial components in Cartesian coordinates, \( \rho \) is the density, \( p \) is the pressure, \( t \) is time, \( \mu \) is the dynamic
viscosity, and \( s_{i,j} \) is the strain-rate tensor. The flow variables \( u \) and \( p \) are then split into their mean and
fluctuating parts as:

\[
u_i = \overline{u_i} + u’_i, \quad p = \overline{p} + p’
\]

where the mean quantities are obtained from Reynolds-averaging, which is a time-average for a flow with
statistically stationary turbulence. In other words, a turbulent flow, whose average does not change with time
[80]. When inserting Equations (3.2) and (3.4) into Equation (3.3) we get:

\[
\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} = -\frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial^2 \overline{u_i}}{\partial x_j \partial x_j} - \frac{\partial u’_i u’_j}{\partial x_j}
\]

The first term is a time-derivative and, therefore, only present for unsteady RANS simulations, for instance,
the simulation of a flow where the mean values decay with time. The term \(-u’_i u’_j = \tau_{ij}\) is the Reynolds
Stress Tensor and needs to be modelled due to a closure problem. The symmetric tensor \( \tau_{ij} \) has six in-
dependent components, hence, six additional unknowns without additional equations to solve them. In a
three-dimensional flow, there are already four mean-flow variables: the pressure, and the three velocity
components. Combined with the additional six components of the Reynolds Stress Tensor, we have ten unknown
properties. The available equations, however, are the mass conservation equation (Equation (3.2)) and three
additional equations of momentum conservation (Equation (3.3)) totalling to only four available equations.
More about how to overcome this closure problem will follow in the next section.

In this thesis, compressible, unsteady RANS simulations will be performed. When simulating a compress-
ible flow, the conservation of energy, Equation (3.8), is added to the set of equations and the density is treated
as time-dependent:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0 \\
\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_j u_i) = \frac{\partial p}{\partial x_i} + \frac{\partial t_{ij}}{\partial x_j} \\
\frac{\partial}{\partial t}\left[\rho \left(e + \frac{1}{2} u_i u_i\right)\right] + \frac{\partial}{\partial x_j}\left[\rho u_j \left(h + \frac{1}{2} u_i u_i\right)\right] = \frac{\partial}{\partial x_j}(u_j t_{ij}) - \frac{\partial q_j}{\partial x_j}
\]

where \( e \) is the specific internal energy, \( h \) is the specific enthalpy, \( t_{ij} \) is the viscous stress tensor, and \( q_j \) is the
heat flux. Additionally, for gases, the ideal gas law is used:

\[
p = \rho RT
\]

The pressure and momentum are solved coupled by the Rhie-Chow algorithm where spatial discretisation
is achieved by second order upwind schemes and a first order implicit transient formulation is used.
3.2.2. Turbulence modelling
Due to the closure problem, either additional model equations addressing each component of the Reynolds Stress Tensor (Reynolds Stress Models) or more general equations modelling the turbulence in the flow (Eddy Viscosity Models), are necessary. Two commonly used Eddy Viscosity Models are the Spalart-Allmaras, and the $k-\omega$ SST turbulence models. However, both models exhibit difficulties in capturing the total pressure gradient in tip vortices because of numerical diffusion [65]. A remedy could be to locally refine the grid. However, this increases the computational cost, and, there may still be a considerable inaccuracy since two-equation models capture poorly the lift and drag of a circulation control wing [39, 62]. The wing in this thesis is not necessarily a circulation control wing as it lacks the ejection of a jet close to the trailing edge. However, an objective of the research still is to find out whether the proposed propeller configuration can induce a Coanda Effect, similar like a circulation control wing, to postpone flow separation and increase lift. A plausible reason for the incorrect calculation of the lift in Ref. [39] is stated to be a too high turbulent kinetic energy which leads to a later flow separation over the flap than in reality and, hence, a larger lift. Since in this thesis both the tip vortices of the propeller blades, and flow separation over the flap are of great importance, it is decided not to use an Eddy viscosity Model but a Reynolds Stress Model.

The turbulence model that will be used, is the $k-\omega$ based Explicit Algebraic Reynolds Stress Model (EARSM) by Wallin and Johansson [76]. The authors claim that the proposed model is numerically robust and only negligibly increases the computational costs compared to two-equation eddy viscosity models by explicitly relating the Reynolds Stresses to the mean flow field. At the same time, the model is supposed to predict effects of rotation, and the separation behaviour of boundary-layers in adverse pressure gradients, better than the two-equation models due to a more realistic representation of the anisotropy in the flow. This will be tested, to some extent, in Section 5 by comparing a boundary-layer profile and pressure distribution of the isolated wing between different turbulence models and the experimental result.

3.2.3. The overset approach
A Chimera, or Overset, grid consists of a background grid, often Cartesian, and overlapping component grids, as shown in Figure 3.5. The outer boundary of the component mesh needs to be defined as overset boundary condition for Fluent to interpret it as a Chimera mesh. In the overlapping area, the solution is either obtained by calculating it or interpolating the results from other overlapping, solved, cells. In Fluent, these solved cells are called donor cells and the information receiving cells are called receptor cells. Eliminated cells, either because they fall outside of the computational domain or because they are replaced by cells of the component mesh, are called dead cells. Due to the interpolation at the boundaries of the component meshes, conservation is not easily enforced [26]. Therefore, it is important that the overlapping cells of the background mesh and the component meshes have a similar size and the zones overlap sufficiently [13]. The advantage of this method is that complex geometries, with multiple components, can be discretised easily with a structured mesh, optimised for each component. Configurations can be changed and parts swapped, without the necessity to create a new overall mesh. Even moving or rotating bodies can be dealt with more easily as the component mesh translates, and rotates, with the body in a stationary background mesh. However, this means that the connectivity between the background and component mesh needs to be re-created after each time step. At the solution initialisation, Fluent completes three main tasks to establish connectivity between the background, and component grids. These are:

![Figure 3.5: Component mesh of a cylinder on a Cartesian background mesh, from Ref. [59].](image-url)
1. **Hole cutting:** Cells that belong to a component mesh but extend outside the domain of the background mesh, and cells of the background mesh that are enclosed by a wall boundary of a component mesh, are cut out and marked as dead cells. In the example of Figure 3.5, dead cells would be found in the top part of the cylinder mesh and in the background mesh where the cylinder is placed. For the propeller-wing mesh, cells will be removed where the propeller mesh intersects with the wing model, and where the wing mesh intersects with the propeller model, as can be seen when comparing Figure 3.6, before the hole-cut, and Figure 3.7, after the hole-cut. The grid after hole cutting has then a maximum mesh overlap. This is visible in Figure 3.7(b) as the area between the red and the green line. The red line frames the outer fringe of the component mesh, that is, the beginning of the overlap area. The green line borders the edge of the hole cut into the background mesh, hence, the end of the overlap area. It can be seen in Figure 3.7, that nearly all the cells of the component mesh overlap with the background mesh. This is computationally inefficient as more cells will be solved than necessary. The receptor cells of the component grid are in the vicinity of the overset boundary condition (red line), and the receptor cells of the background mesh are at the border of the cut-out (green line). In between, the NS equations are solved on both grids. Depending on the refinement of the respective grid, the dimensions of the hole-cut can vary. A coarser mesh usually results in a larger cut-out, if its cells cannot adequately encompass the cut out shape. The aerodynamic performance of the wing due to the hole boundary location can vary slightly [14].

![Figure 3.6: Wing (background) mesh and propeller (component) mesh before hole cutting.](image)

![Figure 3.7: Wing (background) mesh and propeller (component) mesh after hole cutting.](image)

2. **Overlap minimisation:** The overlap is minimised by transforming solve cells into receptor cells and eliminating redundant receptor cells as dead cells. A solve cell is only converted into a receptor cell if a suitable donor cell, with a higher donor priority, can be found. By default, smaller elements have a higher donor priority, such that the solution is obtained on the finest local mesh. As a result, the solution quality is improved by moving the interpolation interface to an area where the grids have a similarly, small, element size. Alternatively, the donor priority can also be specified by the boundary-distance method. The boundary distance method minimises the overlap such that the interpolation area has approximately an equal distance to the wall boundaries of the individual grids. The computational grid after overlap minimisation, for a cell-size based and boundary-distance based donor priority, is shown in Figure 3.8 and Figure 3.9, respectively.
Another way of giving priority to cells during the overlap minimisation is by attributing a higher priority to a whole grid, resulting in the overset mesh in Figure 3.10. A grid with higher priority will almost completely maintain its outer dimensions, while the lower-priority grids minimise their overlap. This prevents spots of local interpolation areas due to an alternation of finer cells between two grids. However, it bears the risk of an inferior match in cell volume if the high priority grid is too fine, or not fine enough. It is recommended to always use a overlap minimisation as it reduces the computational effort and improves convergence [32]. In this study, a higher priority is given to the propeller mesh with the cell-size based overlap minimisation. The reason for giving the propeller mesh a higher priority is to enforce a constant position of the interpolation area with respect to the wing. If both meshes had the same priority, the interpolation area would rotate with the finest cells of the propeller mesh, resulting in an alternating distance between wing and overset boundary, based on the phase angle of the propeller.

Figure 3.8: Overlap minimisation based on matching cell volumes.

Figure 3.9: Overlap minimisation based on boundary distance.

Figure 3.10: Overset interface with a higher priority for the propeller mesh.
3. Donor search: As a last step of creating a overset mesh, Fluent looks for suitable donor cells in one mesh to interpolate the solution of these cells to the receptor cells in the other, overlapping, mesh. The receptor cells are always at the border to the removed, dead, cells. Only solved cells can become donor cells. Valid donor candidates are the cell containing the centroid of the underlying receptor cells, and its adjacent solve cells. The central donor cell is then determined based on the specified overlap-minimisation, for instance, the overlapping solve cell with a volume closest to the receptor cell volume. Figure 3.11 shows schematically, a valid overlap in the gap between the wing and a propeller blade tip. The sketch is meant to highlight the importance of having similar cell volumes in the overlap region, and visualise that there are overlaying solve cells. Hence, flow variables extracted across an overset boundary can have two solutions at some points. A valid receptor cell can have multiple donor cells but must have at least one. Otherwise, it becomes an orphan cell. This might happen if too many, or too few, cells are removed by the hole cutting process, or if there is insufficient overlap across the grids [13]. Gradients for a receptor cell are calculated from the neighbouring solve cells in the same mesh and interpolated from the central donor cell of the overlapping mesh.

Figure 3.11: Donor search in the overlap area between the wing surface and a blade tip.

3.2.4. Computational domain and boundary conditions

There are four main configurations simulated in this thesis. These are: the isolated propeller, isolated wing (with and without propeller-nacelle), baseline over-the-wing configuration, and the inclined over-the-wing configuration. The isolated wing and the baseline OTW configuration are available with, and without, wind tunnel walls, whereas the isolated propeller and the inclined OTW configuration are only modelled without wind tunnel walls.

The computational domain of the isolated propeller simulations is shown in Figure 3.12. The flow is solved compressible, where the working fluid is assumed to be an ideal gas, and Sutherland’s Law is used to calculate the dynamic viscosity. To reduce the computational effort, the calculations are reduced to steady simulations by using a multi-reference frame approach. The walls of the propeller blade, spinner, and nacelle are modelled as no-slip walls with a first layer height below $y^+ = 1$. A total gauge pressure, and total temperature are specified at the inlet whereas at the outlet, the pressure is prescribed to be the ambient, standard, sea-level pressure. Values for the turbulence quantities $k$ and $\omega$ are chosen based on the recommendations by Spalart and Rumsey [64], and source terms are used in the flow to prevent their decay. Only one blade, with periodic boundary conditions at the side walls is modelled in the domain to reduce the required calculation time. At the top boundary a pressure far-field boundary condition is used with prescribed Mach number, static pressure, and static temperature according to the inlet condition.

The isolated wing and its domain are exactly identical to either one of the two OTW propeller domains, except for the volume where the propeller mesh is added with the overset technique. Therefore, the domain with wind tunnel walls is shown as an example for the baseline OTW configuration in Figure 3.13. The cross-section shown in the top of the figure has the same dimensions as the cross-section of the test-section used in the wind tunnel experiments for validation. The domain extends approximately 3 chords upstream and 5 chords downstream of the wing, where it was ensured that the inlet and outlet are sufficiently far away from
the wing by evaluating their influence on the wing for varying distances. The inlet and outlet boundary conditions, the source terms in the flow, as well as the ideal gas law for the equation of state and Sutherland’s Law, are used similar to the isolated propeller simulations. The sidewalls of the wind tunnel intersecting with the wing model are set to a no-slip boundary condition for the development of a boundary-layer on these walls. To reduce the computational effort, however, a wall functions approach is used at these sidewalls contrary to the resolved boundary-layer with $y^+ < 1$ on the wing. The other walls of the wind tunnel are set as free-slip walls by prescribing zero wall shear. It should be noted that only the effective cross-section of the test-section is modelled since the real test-section from the experiments has diverging walls such that the boundary-layer edge on the sidewalls approximately leads to a constant-area test section to prevent boundary-layer blockage. Therefore, the boundary-layer imposed on the modelled sidewalls that is necessary to account for the aerodynamic interaction with the boundary-layer on the wing, causes a small blockage effect.

To remove the influence of the wind tunnel walls, the sidewalls are changed to symmetry boundary conditions and the top and bottom of the domain are extended with far-field pressure boundary conditions ten
chord lengths away, as shown in Figure 3.14. The prescribed quantities in the flow, at the inlet and outlet, as well as the wing and propeller mesh, are not changed. These boundary conditions are used for both the baseline and inclined OWT propeller. The inclined OTW propeller position is derived from the baseline configuration by inclining the propeller around the flap hinge by the angle of flap deflection, $\delta_f$, as shown in Figure 3.15. Consequently, the inclined propeller is positioned behind the flap curvature, closer to the trailing edge than the baseline propeller, and the axis of the inclined propeller is parallel to the upper flap surface. This configuration represents a propeller as if it were physically attached to the flap with the purpose of postponing flow separation on the flap by blowing a high-momentum flow over it.

Figure 3.14: Computational domain of the inclined configuration without wind tunnel walls.

Figure 3.15: Position of the inclined over-the-wing propeller.
In this chapter, the iterative and discretisation error will be estimated for the isolated wing, and propeller. Then, the influence of the overset interface on the boundary-layer of the wing, and the sectional lift and drag coefficients will be studied. The model error of the isolated components and of the integrated system will be evaluated in Chapter 5 and Section 7.1, respectively, by validating the numerical results with the wind tunnel measurements.

4.1. Error estimation and accuracy

There are three main errors in computational modelling: the modelling error, the discretisation error, and the iteration error. Ideally, the respective error should reduce one order of magnitude from modelling, to discretisation, to iteration error for the individual error estimation to be accurate [26]. These errors and their sources are described in the following:

**Modelling error:** Solving the NS equations exactly, is only possible for very limited, simple, flow cases. Therefore, modelling is necessary that introduces an error compared to the exact NS solution. In case of a turbulent flow, turbulence models are indispensable for RANS simulations. No turbulence model is perfect, as they are only approximations, and thus, they contribute to the modelling error. Similarly, approximations such as incompressible flow can influence the modelling error, even at low Mach numbers. Furthermore, no simulation is run in an infinite space. Instead, artificial boundary conditions are used at the inlet, outlet, and lateral borders of the finite computational domain. Finally, not always can all details of the real geometry be discretised in the computational domain. Very small structures are sometimes neglected and round surfaces become edged depending on the discretisation density. Consequently, even if the mathematical model is solved correctly, there is always a discrepancy to the real flow, called the modelling error.

**Discretisation error:** The governing equations are differential equations that have to be discretised in time and space on a grid of discrete points. The resulting algebraic system of equations can then be solved by the computer. Again, an approximate solution is produced; however, this time the solution of the previously established model is approximated by a solution at each discrete grid point. The order of an approximation describes its quality by relating the truncation error to an exponent. For instance, an approximation of the order $p$ means that the truncation error of a spatial derivative is proportional to $(\Delta x)^p$, where $\Delta x$ is a characteristic grid spacing. Hence, for a second order approximation, the truncation error is reduced by a factor 4, if the grid spacing is refined (reduced) by a factor 2. The order $p$ is also called Grid Convergence Index [26]. The accuracy of the solution can be increased by using better approximations, i.e. increasing the temporal and spatial grid resolution. However, this also increases the required resources, since the computational cost is proportional to the number of grid elements. Therefore, it is beneficial to find a grid resolution which results in an acceptable accuracy at an as low as possible cost. Furthermore, not only the size of the grid elements influence the computational efficiency, but also their distribution. Locations with strong flow gradients require a higher grid density than areas of smaller change in the flow variables. If the grid resolution in the direction of the flow gradient is not high enough, the change of the flow variable from one element to another is large, and so is the discretisation error.
**Iteration error:** The solution of the discretised equations cannot be obtained exactly, because, after a determined amount of iterations the computation has to be stopped. This is usually done by defining a convergence criterion. Even if the calculations would run indefinitely, the error between the exact solution of the discretised equations and the iterative solution, would decrease but never become zero, due to the precision of the computer. Calculations done with double precision are, for most cases, accurate enough so that the round-off error of the computer becomes negligible and the iterative error can be reduced to a value small enough compared to the other errors [26]. Again, it is necessary to find an optimum between the number of iterations and the accuracy of the result, such that the computational cost can be kept minimal.

The error estimations will be done for four systematically refined grids. To obtain a next finer mesh, a grid refinement ratio of 1.3 is used. The grids are named $h_1$ to $h_4$ where $h_1$ is the finest one. Only the first-layer height and number of layers in inflation layers were kept constant. The wing and propeller are considered individually, instead of as combined OTW system, to save computing time and to ensure that the forces of the individual components are captured accurately, before combining them. An uncertainty estimation, due to the discretisation error, will be given by using Eça and Hoekstra’s procedure [22].

### 4.1.1. Isolated Wing

To estimate the iterative error, the lift and drag coefficients of the wing profile are plotted per iteration in Figure 4.2. It can be observed that already after 200 iterations steady values are reached. Also the velocity magnitude in the point where the center of the propeller spinner will be was monitored (not shown here), and exhibited the same behaviour. However, the simulations were run for 1000 iterations in total, for all the residuals to drop at least five orders of magnitude. The change in lift and drag per iteration after the full 1000 iterations is less than 0.1%. Therefore, the iterative error is negligibly small. This will be confirmed by comparing it to the discretisation error.

![Figure 4.1: Lift coefficient per iteration.](image1)

![Figure 4.2: Drag coefficient per iteration.](image2)

The discretisation error $\epsilon$ is calculated by a power series expansion as:

$$\epsilon = \alpha h_i^p$$

(4.1)

where $\alpha$ is a constant to be determined, $h_i$ is the representative grid cell size, and $p$ is the observed order of grid convergence. The assumptions inherent to this procedure are that the grids are in the so-called asymptotic range and geometrically similar. Since these assumptions are rarely completely true in practical applications, Eça and Hoekstra provided two modifications to Equation (4.1) which can be found in Ref. [22]. From Figures 4.3 and 4.4 it can be inferred that the lift and drag are monotonically converging with an observed order of convergence in the range $0.5 < p < 2$, that is the range for which the error estimation is considered reliable [22]. Hence, Equation (4.1) can be used to estimate the discretisation error. The fit through the data points provides the theoretical solution for a mesh with infinite number of grid points ($h_i/h_1 = 0$). The representative grid cell size $h_i$ is calculated from the total volume $V$, and the number of cells $n$, of each grid as:

$$h_i = (V/n_i)^{1/3}.$$ 

The discretisation error is then the difference between the theoretically correct solution at $h_i/h_1 = 0$ and the actual solution at the respective grid cell size. It can be seen in Table 4.1, that the magnitudes of the errors $\epsilon_{CI}$ and $\epsilon_{Cd}$ decrease with increasing refinement, as desired. Mesh $h_3$ is discretised such...
that with a time step equivalent of two degrees of propeller rotation in the combined simulations, the flow travels approximately one cell in the wing mesh at the propeller location. The other levels of mesh refinement are derived from the grid $h_3$. Therefore, the sizing of this grid already results in a relatively large number of cells and will be used for the combined OTW propeller simulations. Moreover, Figure 4.5 shows that the velocity profiles at the propeller location deviate only marginally in maximum velocity from the finest mesh to the grid $h_3$. The uncertainty in the lift and drag coefficient is also shown in Table 4.1 and calculated from the discretisation error, the standard deviation of the data points $\sigma$, and the deviation of the data points from the fit as:

$$U_{C_l} = F_s \epsilon_{C_l} + \sigma + |C_{l,i} - C_{l,fit}|$$

where $F_s$ is a safety factor based on the grid convergence. The uncertainty for the drag coefficient follows the same procedure and both represent a range that contains the exact solution with 95% coverage.

### Table 4.1: Error and uncertainty of the lift and drag coefficients in the units of the respective coefficients.

<table>
<thead>
<tr>
<th></th>
<th>$h_1$</th>
<th>$h_2$</th>
<th>$h_3$</th>
<th>$h_4$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\epsilon_{C_l}$</td>
<td>-0.0029</td>
<td>-0.0037</td>
<td>-0.0046</td>
<td>-0.0056</td>
</tr>
<tr>
<td>$U_{C_l}$</td>
<td>0.0039</td>
<td>0.0048</td>
<td>0.0062</td>
<td>0.0074</td>
</tr>
<tr>
<td>$\epsilon_{C_d}$</td>
<td>+0.000197</td>
<td>+0.000270</td>
<td>+0.000369</td>
<td>+0.000498</td>
</tr>
<tr>
<td>$U_{C_d}$</td>
<td>0.000338</td>
<td>0.000403</td>
<td>0.000526</td>
<td>0.000715</td>
</tr>
<tr>
<td>Cells</td>
<td>208'212</td>
<td>128'342</td>
<td>79'404</td>
<td>50'224</td>
</tr>
</tbody>
</table>

### Figure 4.3: 2D lift coefficient refinement.

### Figure 4.4: 2D drag coefficient refinement.

### Figure 4.5: Non-uniform axial velocity distribution along a vertical line at the chordwise propeller location.
4. Verification

4.1.2. Isolated propeller

As for the isolated wing, the isolated propeller simulations were run until the iterative error of the thrust and torque coefficients was less than 0.1% and the residuals dropped at least five orders of magnitude. This was achieved after 5000 iterations as shown in Figures 4.6 and 4.7. The thrust coefficient converged with an observed order of grid convergence $p$ in the acceptable range and is shown in Figure 4.8. However, the observed order of grid convergence for the torque coefficient was too large ($p > 2$) which can result in too small error estimates. Therefore the respective error is calculated as $\epsilon_{CQ} = \alpha h_p^{p}$ according to Eça and Hoekstra’s procedure [22]. The resulting discretisation errors and uncertainties are shown in Table 4.2.

![Figure 4.6: Thrust coefficient per iteration.](image)

![Figure 4.7: Torque coefficient per iteration.](image)

![Figure 4.8: Thrust coefficient refinement.](image)

![Figure 4.9: Torque coefficient refinement.](image)

<table>
<thead>
<tr>
<th></th>
<th>$h_1$</th>
<th>$h_2$</th>
<th>$h_3$</th>
<th>$h_4$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\epsilon_{CT}$</td>
<td>+0.0021</td>
<td>+0.0028</td>
<td>+0.0036</td>
<td>+0.0047</td>
</tr>
<tr>
<td>$U_{CT}$</td>
<td>0.0028</td>
<td>0.0037</td>
<td>0.0047</td>
<td>0.0060</td>
</tr>
<tr>
<td>$\epsilon_{CQ}$</td>
<td>-0.00022</td>
<td>-0.00035</td>
<td>-0.00053</td>
<td>-0.00078</td>
</tr>
<tr>
<td>$U_{CQ}$</td>
<td>0.0008</td>
<td>0.0012</td>
<td>0.0017</td>
<td>0.0025</td>
</tr>
<tr>
<td>Cells</td>
<td>14'463'300</td>
<td>7'346'487</td>
<td>3'909'325</td>
<td>2'192'981</td>
</tr>
</tbody>
</table>

Table 4.2: Error and uncertainty of the thrust and torque coefficients in the units of the respective coefficients.

Figure 4.10 shows no clear trend in the convergence of the thrust distribution of the propeller blade for the four grids. Figure 4.11, however, shows that at a distance to the propeller tip equal to the tip clearance used in the OTW configuration, only mesh $h_4$ shows a clear offset in axial velocity in the region influenced...
by the blade tip vortices. The grids $h_2$ and $h_3$ show an axial velocity similar to each other in this region, and therefore, mesh $h_3$ will be used to keep the number of cells low, while the discretisation error and uncertainty of the thrust coefficient is in the order of 1%.

4.2. Influence of overset modelling

To study the influence of the overset (OS) interface on the lift and drag of the wing, as well as on the boundary-layer at the propeller location, simulations were run with the wing and an "empty" overset mesh, i.e. without the propeller geometry in it. Thereby, it became obvious that the mesh overlap needed to be improved to prevent the solver from diverging. Therefore, the overset mesh used for the OTW propeller configurations features a structured mesh on the outside of the overset cylinder for improved overlap with the structured wing mesh, as shown in Figure 4.12.

Since the propeller tip clearance is relatively small, overlapping solve cells from the background and component mesh can be found close to the surface of the wing. This is visible as spikes in the boundary-layer profile in Figure 4.13 (a). The volumes of the cells in the structured part of the overset mesh are defined such that the overset interface is created approximately in the middle of the tip gap with the cell-size based overlap minimisation technique. The overset interface results in a local kink in the boundary-layer profile at a distance of approximately $d/c = 0.002$ to the wing-surface. At this location the mesh of the wing and the propeller are
merged and errors in the flow gradient may arise due to the interpolation across the interface. Close to the wing surface, the overset interface can cause an increase in wall shear due to an increased vertical velocity gradient. It can be inferred from Table 4.3, however, that this has a negligible influence on the boundary-layer thickness $\delta_{99}$. Furthermore, to understand how the influence of the overset interface compares to the effect of modelling the wing in 2D or 3D, the value for the boundary-layer from the two-dimensional results is given as well.

The pressure distribution of the wing profile, shown in 4.13(b), is affected only slightly, on the upper surface. The suction peak is higher with the overset interface than without, due to a locally higher dynamic pressure caused by the overset interface. This results in a sectional lift coefficient in the middle of the wing that is $\Delta C_l = 0.01$ higher with the overset interface than without, as shown in Table 4.3. The total sectional drag coefficient, however, appears practically unaffected by the overset interface due to a small decrease in pressure drag but similarly small increase in friction drag.

![Figure 4.13: Boundary layer profile](image)

![Figure 4.14: Pressure distribution](image)

**Figure 4.13:** Boundary layer profile of the wing at the propeller location and pressure distribution at the half span of the wing compared between the isolated wing without an empty propeller domain, and the wing with an empty, non-rotating, overset domain.

**Figure 4.14** shows the pressure coefficient on the wing and streamlines following the local shear stresses below the empty overset cylinder. It can be seen that the stronger flow gradient close to the wall caused by the overset interface results in an apparent postponed flow separation in the middle of the wing. That is why the sectional lift and friction drag coefficients increase, whereas the pressure drag coefficient decreases compared to the 3D case without overset interface. The resulting model error caused by the overset interface is given in Table 4.3. Additional results showing good and bad cell overlap are given in Appendix B. In the following section, the model error of the isolated wing, without overset interface, will be assessed by comparing the numerical results to the experimental ones.
Figure 4.14: Influence of the overset interface on the shear stresses on the wing.
This section covers the validation of the isolated wing, without an overset interface, by comparing the numerical results to the ones obtained from wind tunnel tests. This is done separately from the validation of the OTW system in Section 7.1 to make sure that the isolated wing is modelled accurately, and possible deviations compared to the experiment of the combined system are due to a poor capturing of the aerodynamic interaction effects, e.g. propeller induced flow separation. First, the boundary-layer of the wing, at the propeller location, will be compared between different turbulence models and the experimental results. Thereby, a benchmark is given for how well the EARSM model compares to other $k-\omega$ based models. Then, the pressure distributions, and the resulting lift coefficients, of the wing profile obtained from the numerical results are compared to the experimental results. Finally, the flow field over the flap is validated by comparing the axial velocity and turbulent kinetic energy obtained from the CFD simulations to the PIV results.

5.1. Isolated wing boundary-layer profile

The boundary-layer profiles are given in Figure 5.1. Besides the profile obtained from the Reynolds Stress model (EARSM), boundary-layer profiles obtained with the Standard $k-\omega$ model [81], Baseline $k-\omega$ model (BSL) [44], and the Shear-Stress-Transport $k-\omega$ model (SST) [44] are shown. The results are obtained from 2D simulations but for comparison, the values from the 3D simulations with the EARSM model are given as well. All the profiles are similar in shape and boundary-layer thickness. The largest deviation is found for the Standard $k-\omega$ model. The corresponding values of the boundary-layer thicknesses are listed in Table 5.1. It can be expected that the BSL and SST models perform better than the Standard model since they are derived from the Standard model with a reduced dependency of free-stream values, and especially for the SST model, an improved accuracy for flows with an adverse pressure gradient [44]. It is somewhat disappointing, however, that the Reynolds Stress model delivers no additional improvement in accuracy over the Baseline model, with the SST model actually being closest to the experimental results. Nevertheless, an error of 3% for such a thin boundary-layer is acceptable.

<table>
<thead>
<tr>
<th>Experiment</th>
<th>Standard</th>
<th>BSL</th>
<th>SST</th>
<th>2D EARSM</th>
<th>3D EARSM</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta_{99}/c$</td>
<td>0.0116</td>
<td>0.0132</td>
<td>0.0119</td>
<td>0.0116</td>
<td>0.0119</td>
</tr>
<tr>
<td>$\Delta\delta_{99}/c$</td>
<td>-</td>
<td>+14%</td>
<td>+3%</td>
<td>0%</td>
<td>+3%</td>
</tr>
</tbody>
</table>
5. Validation

Figure 5.1: 2D boundary layer profiles for different $k - \omega$ turbulence model options.

5.2. Isolated wing pressure distribution

An improvement in accuracy by the SST and the EARSM becomes evident when the pressure distributions are compared in Figure 5.2. The SST and EARSM model show a better agreement in the pressure at the trailing edge compared to the pressure distribution obtained from the experiments, which is attributed to their superior performance for flows with adverse pressure gradients. Consequently, the suction peak is not overpredicted, as it is the case with the Standard and BSL models, and the resulting lift coefficient is closer to the one obtained from the experiments. The lift coefficients are calculated by exporting the pressures shown in Figure 5.2 and integrating them with the same Matlab script as for the experimental results to reduce additional errors due to the calculation procedure. In Table 5.2, it can be seen that the absolute differences in lift coefficients compared to the experimental result are low, however, relative to the already very low lift coefficient of the wing they can become significantly large. Therefore, the EARSM proves to be the best choice of $k - \omega$ models as it results in a lift coefficient closest to the one from the wind tunnel experiments. Results obtained from $k - \epsilon$ models are not shown here as they exhibited even larger errors in preliminary simulations, due to the improved capability of $k - \omega$ models to predict boundary-layer flows [44].

Table 5.2: Lift coefficients for different $k - \omega$ turbulence model options.

<table>
<thead>
<tr>
<th></th>
<th>Experiment</th>
<th>Standart</th>
<th>BSL</th>
<th>SST</th>
<th>EARSM</th>
<th>3D EARSM</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_l$</td>
<td>0.097</td>
<td>0.173</td>
<td>0.209</td>
<td>0.068</td>
<td>0.099</td>
<td>0.100</td>
</tr>
<tr>
<td>$\Delta C_l$</td>
<td>-</td>
<td>+0.076</td>
<td>+0.112</td>
<td>-0.029</td>
<td>+0.002</td>
<td>+0.003</td>
</tr>
</tbody>
</table>

Figure 5.2: Pressure coefficient distribution along the airfoil chord for different $k - \omega$ turbulence model options.
To compare the flow field from the numerical results to the averaged PIV results, the spinner and nacelle were added to the isolated wing. Since the previous evaluation of the pressure distribution showed the best agreement with the experiments for the EARSM, only this model is used for the comparison to PIV. The flow fields are compared in Figure 5.3, where no significant difference can be seen between the numerical and the experimental results. Only the velocity profiles along a vertical line at $x/c = 0.9$ reveal slightly lower velocities from CFD than PIV. This can be attributed to an earlier separation in the numerical results compared to the experimental ones due to a lower turbulent kinetic energy close to the wing surface, as shown in Figure 5.5. However, the difference is small and spurious vectors in the results of the turbulent kinetic energy in Figure 5.5(a) indicate an inaccurate detection of the displacement of a set of particles in the flow. Hence, with the uncertainty in the values obtained from the experiments taken into account, the numerical results of the isolated wing are sufficiently accurate. Nevertheless, velocities in the boundary-layer over the flap are underpredicted for the isolated wing by the numerical simulation compared to the wind tunnel measurement.

![Figure 5.3: Comparison between the CFD RANS results and the PIV measurement for the velocity component in the x direction over the flap.](image)

![Figure 5.4: Axial velocity profile at the axial coordinate $x/c = 0.9$. Errorbars indicate the uncertainty in the velocity obtained from PIV.](image)
Figure 5.5: Comparison between the CFD RANS results and the PIV measurement for the turbulent kinetic energy of the flow over the flap.
III

Results
Aerodynamic characteristics of individual components

Before analysing the results of the complete OTW-system, it is useful to understand the aerodynamic characteristics of the isolated wing and propeller. The following two sections will highlight important aspects of the flow field around the isolated wing and the propeller to get a preliminary understanding of how they will influence each other. The results of the wing include wind tunnel wall-effects because their influence will be discussed later. The propeller simulations are performed without walls, using periodic and far-field boundary conditions instead. Including the walls in the propeller simulations, would require a complete six-blade model of the propeller inside the wind tunnel section. This would increase the computational time significantly, for a negligible increase in accuracy compared to the experiments, as will be shown in Section 7.2 on the influence of the wind tunnel walls.

6.1. Isolated wing

The wing creates relatively low lift ($C_L = 0.1$) due to its almost symmetrical profile and zero angle of attack. Therefore, also the velocity increase around the wing, and the curvature of the surrounding streamlines, is low, as can be seen in Figure 6.1. Nevertheless, the downwash of the wing deflects the streamlines slightly, such that the baseline propeller will be at a small negative angle to the incoming flow. The angle is defined negative as counter-clockwise from the streamline to the propeller axis, and positive in the other direction, as sketched in Figure 6.2. The inclined propeller is rotated around the y-axis more than the streamlines are deflected downwards, hence, it is at an positive angle. Both the baseline and inclined propeller will operate in an area of increased inflow velocity compared to free-stream. However, the baseline plane is upstream of the inclined position, where the local flow acceleration is maximum. It should be kept in mind, that this is only the isolated wing, and that propeller induced flow separation will likely affect the flow acceleration and streamline curvature. Separated flow will cause a weaker acceleration around the flap curvature and less streamline curvature. However, the flow field around the isolated wing can still be used to explain the non-uniform inflow into the propeller.

The relative inflow velocities into the respective propeller planes are shown in Figure 6.3. The propeller is not present, only the nacelle and spinner. For the inclined plane, the velocity component perpendicular to the plane is shown in a coordinate system rotated with the propeller as:

$$u = V_x \cos(\delta_f) - V_z \sin(\delta_f)$$

(6.1)

where $\delta_f = 20^\circ$ is the deflection of the flap. Iso-lines of the change in local blade angle of attack, $\Delta \alpha$, due to the non-uniform inflow are superimposed on the inflow-contours. The change in angle of attack is calculated in the baseline plane as:

$$\Delta \alpha = -\Delta \Phi = -\tan^{-1}\left(\frac{V + \Delta V_x}{\Omega r + \Delta V_t}\right) - \tan^{-1}\left(\frac{V}{\Omega r}\right)$$

(6.2)

where the change in tangential velocity is defined as $\Delta V_t = \Delta V_y \cos(\phi) - \Delta V_z \sin(\phi)$, the angle $\phi$ is the phase angle, and the angle $\Phi$ is the angle between the effective inflow velocity and the tangential velocity. In other
words, $\Phi$ is the pitch angle ($\beta$) of the blade minus its angle of attack ($\alpha$), as was defined in Figure 2.1. In the inclined plane, $\Delta \alpha$ is calculated from the respectively rotated velocity components. It should be noted, however, the influence of the velocities induced by the propeller are not included.

The change in blade angle of attack in the baseline plane reveals two non-uniformities of the inflow and their relative contribution. Firstly, the higher velocities close to the wing, at $\phi = 0^\circ$, increase the local advance ratio. Hence, the blade angle of attack is decreased compared to the isolated propeller, and also compared to phase angles in areas of slower inflow, for instance at $\phi = 180^\circ$. Secondly, the negative angle of the streamlines to the propeller plane increase the tangential velocities for the up-going blades ($0^\circ < \phi < 180^\circ$), hence increase the loading locally. This should result in an increase in blade angle of attack for the up-going blades. However, as can be seen in Fig. 6.3(a), a negative change in angle of attack, with respect to an uniform inflow, can be found almost every radial and azimuthal coordinate. Hence, the decrease in $\alpha$, due to the higher inflow velocity, is larger than the increase in $\alpha$, due to the locally increased tangential velocity. For the down-going blades ($180^\circ < \phi < 360^\circ$), the tangential velocities decrease while the axial inflow velocities stay the same, compared to the up-going blades. Hence, the relatively large decrease in angle of attack. A similar pattern in change of local advance ration was also found in simulations of the same wing and propeller but at twice the free-stream velocity in Ref. [70].

In the inclined plane, the inflow velocities are lower due to the contribution of $\cos(\delta_f)$ on $V_x$. However, the influence of the wing is still present, especially in the range of $300^\circ < \phi < 60^\circ$. The down-going blades experience a larger tangential velocity now, due to the positive angle between the streamlines and propeller plane. This change in tangential velocity is significantly larger than the one for the baseline plane, such that the blade angles of attack increase for the down-going blades ($\phi = 270^\circ$) and decrease for the up-going blades ($\phi = 90^\circ$). Close to the wing the angles of attack still decrease, however, less than in the baseline case.

Since the previous results include the nacelle spinner, their influence on the pressure distribution of the wing profile will be discussed briefly. The nacelle increases the blockage, hence velocities increase between the wing and nacelle. This can be seen as an increase of the suction peak for the baseline case in Figure 6.4(a). The spinner of the inclined nacelle is rotated away from the suction peak but the nacelle itself is closer to the flap. Consequently, there is no increase in velocity, hence suction, over the curvature of the flap anymore but over the flap itself. This decreases the adverse pressure gradient locally and leads to flow separation further downstream than with the baseline nacelle, and also the isolated wing, as can be observed in Figure 6.4(b).
6.2. Isolated propeller

The propeller accelerates the flow and decreases the static pressure in front of it, and increases static pressure behind it. Figure 6.6 shows how the axial velocity and the pressure coefficient change along horizontal lines taken at increasing radial distances as indicated in Figure 6.5. The purpose of this analysis is to see how the wing might be affected by a propeller in close proximity. These results are for the isolated propeller operating in a uniform flow field. However, although the absolute values might differ once the propeller is placed in the non-uniform flow above the wing, the trends will likely stay the same. The closest distance to the propeller tip \( \epsilon / R = 0.0738 \), is the same tip clearance the propeller will have once it is over the wing. At this distance, distinct fluctuations in axial velocity and pressure coefficient can be observed due to the blade tip vortices. These oscillations fade away with radial distance and are not present anymore at a distance of half the radius. The blade tip vortices are indicated by the circles at a radial coordinate of \( y' / R = 0.9 \) in Figure 6.5. At \( c / R = 0.0738 \), the axial velocity increases by a maximum of 3% at 0.35\( R \) ahead of the propeller and the pressure coefficient decreases by 0.23 just behind that. This decrease in pressure enhances the suction, hence lift, on the wing once the propeller is close enough to it. It can also be inferred that this is only a local effect as the increase in axial velocity, and suction, decrease quickly in front of the propeller. At 4\( R \) ahead of the propeller, the static pressure recovered to the same level as a wing that is one radius away from the propeller. The
leading edge of the wing in the following study is about eight times the radius ahead of the propeller. Hence, only a small increase in leading-edge suction is expected.
Baseline over-the-wing configuration

In this chapter, the results of the baseline OTW configuration, thus, horizontal propeller alignment, will be used to analyse the capability of the RANS simulation to capture propeller induced flow separation on the wing, study the aerodynamic interaction effects between wing and propeller, and evaluate how the forces of such an OTW system are affected by the aerodynamic interaction. First, the results will be compared to the experimental data. Then, the influence of the wind tunnel walls on the aerodynamics of the OTW system will be discussed by comparing the numerical results to the same OTW configuration but without walls. After the contribution of the wind tunnel walls is understood, the aerodynamic interaction characteristics between the wing and the propeller will be presented for the configuration without wind tunnel walls. Finally, the influence of the wing on the propeller loading, and vice versa, will be discussed.

The following results are all obtained from unsteady, compressible, RANS simulations, unless stated differently. The steady flow fields and surface contours were obtained by phase averaging the unsteady results over 30 time steps, equivalent to 60 degrees of rotation in total. For the six-bladed propeller used in this study, this means averaging from the moment one blade passes the wing perpendicularly, to the next perpendicular blade passing.

7.1. Comparison to experimental results

While running the calculations, not only the residuals were monitored to judge convergence but also the lift and drag of the wing, and the thrust of all blades combined as well as the thrust of one individual blade. The residuals dropped to between $10^{-5}$ (continuity) and $10^{-10}$ (energy) per time step. The convergence of the lift and drag of the wing is shown in Figure 7.1(a). It can be seen that the lift oscillates with a decreasing amplitude and approximately steady wavelength of one propeller revolution. Although the maximum value of lift still decreased, it was decided to stop the simulation, since already 67800 iterations passed, which corresponds to eleven propeller revolutions. The change in lift from revolution ten to eleven is approximately $\Delta C_L = 0.002$ ($\approx 1\%$), while the change in drag is less than 0.1%. However, the maximum change in lift within that propeller revolution is $\Delta C_L = 0.012$. This change is comparable to the corresponding uncertainty due to the grid discretisation established in Section 4.1.1. The propeller rotated two degrees per time step, except for revolution six to seven, for which the time step was decreased to one degree rotation. This had a negligible impact on the lift and drag calculation. Fluctuations in the lift of the wing can be expected due to the passing of the blades. However, with a six-bladed propeller, a peak should appear every 60 degrees of rotation instead of every full propeller revolution as seen in Figure 7.1(a). Therefore, it can be assumed that the oscillations in lift are not due to the blade passing. Nevertheless, they are triggered by the propeller, since they did not occur in the isolated wing simulations. A simulation of the full wing with an identical setup and transient time step but without the propeller was run for comparison, and showed no oscillation in the lift. Interestingly, with the propeller rps of 87 Hz, half the wing thickness of 0.055 m, and the free stream velocity of 20 m/s, a Strouhal number of about 0.2 is obtained. Half of the wing thickness was used as this is approximately the thickness of the wing profile where the flow separates. Over a wide range of Reynolds numbers, the critical Strouhal number for vortex shedding behind a cylinder is about 0.2. Something similar might be happening behind the wing due to its mostly symmetric shape and relatively blunt aft-section. Further investigation of the flow structures behind the wing, confirmed the presence of vortex bands parallel to the trailing edge (see Figure...
C.1 in the appendix). This further supports the assumption that a vortex shedding mechanism causes the alternating increase and decrease in lift. A deeper study of this phenomena, also focusing on the influence of the wind tunnel walls, is necessary to explain why the propeller triggers these oscillations in lift.

Results were saved as backup every other full propeller rotation. This enables the comparison of the velocity profiles of the inflow to the propeller as the simulation advances. The axial velocity is extracted along a vertical line at one radius in front of the propeller, as shown in Figure 7.2. The corresponding velocity profiles are plotted in Figure 7.1(b). Unfortunately, results were not saved after the same number propeller rotations had passed each time, which makes the convergence of the profile less obvious. However, the profiles change only close to the wing, where the maximal axial velocity decreased by only 0.25% from revolution seven to eleven.

For the present study, the lift and drag averaged over the last two propeller revolutions are taken as steady values, which leads to a uncertainty in lift of $U_{CL} = \pm 0.006$ due to the spread between the minima and maxima in lift. The averaged lift and drag coefficients are:

$$C_L = 0.143 \quad (7.1)$$
$$C_D = 0.0143 \quad (7.2)$$

which leads to a lift-to-drag ratio of $L/D = 10$. Compared to the isolated wing ($C_L = 0.10$, $C_D = 0.015$), this corresponds to an increase in lift of 40% and a decrease in drag of 7%. Consequently, the lift-to-drag ratio
increased by almost 50%. In the following comparison, the two-dimensional lift coefficient, $C_l$, will be compared since no three-dimensional lift coefficient is available from the experimental results. First, however, the pressures of the wing will be compared.

Averaging the wing-pressures from CFD over 30 time steps results in the pressure distribution shown in Figure 7.3. Since in the wind tunnel experiment the wing surface pressure could only be measured for one wing profile, the propeller was traversed in the spanwise direction to emulate having a 3D distribution of pressure ports in the middle of the wing. The zig-zag lines in Figure 7.3 represent the coverage of the 3D measurement field from the experiment. As can be expected, the propeller, positioned at $x/c = 0.79$, locally increases the averaged suction on the wing. The resulting bump in the suction over the flap hinge extends to about $x/c = 0.6$ in chordwise direction and $y/c = \pm 0.2$ in spanwise direction.

The resulting change in pressure, $\Delta C_{p, \text{avg}}$, on the suction side of the wing, is calculated as propeller-on pressures ($C_{p, \text{on}}$) minus the propeller-off pressures ($C_{p, \text{off}}$). The propeller-off simulation was run with the nacelle and a dummy-spinner without blades to isolate the effect of blade thrust. In Figure 7.4, the resulting change in pressure is compared between CFD and the experiment. The CFD-pressures were extracted at the same points as in the experiments and plotted with the same Matlab script to obtain a similar resolution. It can be observed that in the experiment the static pressure just behind the propeller increases more than in the simulation. Moreover, the increased pressure area over the flap is followed by a low-pressure area in the experiment, whereas in CFD the deltas at the trailing edge are positive. Furthermore, the increased suction area in front of the propeller is stronger in CFD than in the experiment. The same observations become evident when the pressure difference is compared for the suction side of the wing profile directly below the propeller axis, as shown in Figure 7.5. This is attributed to propeller-induced flow separation over the flap happening more upstream in the experiment than in CFD. This leads to a stronger decrease of the suction peak in the experiment, hence, explain the larger pressure increase by the propeller behind $x'/R = 0$, in the experiment than in the CFD simulation. The later flow separation in the numerical results causes a larger flow acceleration over the flap curvature decreasing the pressure in front of it. Hence, there is larger suction ahead of the propeller observed in the numerical results than in the experimental ones.

The comparison of the flow-field over the flap, in Figure 7.6, and the velocity profiles extracted at three locations along the flap, in Figure 7.7, confirm that in the numerical solution the flow is more aligned with the flap than in the experiment. This discrepancy in separation behaviour can be due to an anomaly in the exper-
Figure 7.4: Comparison between the pressure deltas from the traversing pressure measurement in the experiment (a), and CFD (b). Pressures from CFD were extracted at the same points as in the experiments. The colourbars are not saturated for improved comparability.

Figure 7.5: Pressure difference between propeller-on and propeller-off on the suction side of the wing at $y/c = 0$. The axial coordinate $x'$ has its origin at the propeller location.

ments, an inaccuracy of the CFD simulation, or a combination of both. The wing model for the wind tunnel experiments had surface imperfections due to the manufacturing process, the holes for the pressure ports, and the slit for the PIV laser. All of these deviations from an ideal surface can weaken the boundary layer, which is then more prone to separate when the propeller is present. However, for a correct prediction of flow separation it is crucial to accurately capture the small turbulence scales close to the wall as that is where the separation starts. Since in RANS simulations all turbulence is only modelled, and not resolved, the correct prediction of flow separation strongly depends on the turbulence model that is used. The flow over the isolated wing separates slightly earlier than in the experiment (Sec. 5.3), due to a lower turbulent kinetic energy, but the results of the installed OTW configuration suggest the opposite. However, the pink lines connecting locations of zero axial velocity in Figure 7.6 indicate that flow reversal on the flap starts in both the numerical and the experimental results at approximately $x/c = 0.84$, where the point of flow separation actually appears to be slightly more upstream in the numerical results than in the experimental ones. Nevertheless, the flow follows the flap curvature more when calculated in CFD than measured experimentally. This is attributed to a weaker presence of the blade tip vortices in the numerical results than in the experiment, as shown in Figure 7.8. A strong reduction in vorticity of the numerical results compared to PIV can be found for the second and third tip vortices at $x/c = 0.84$ and $x/c = 0.88$, respectively. The lower vorticity of the blade tip vortices in the numerical results leads to a weaker additionally induced adverse pressure gradient than in the experiments. This is attributed to numerical diffusion which is typical for RANS simulations [65], but is also intensified by the change from the unstructured to structured propeller mesh (Fig. 4.12). To prevent numerical diffusion
and more accurately capture the vortex strength, very fine grids are necessary, requiring an excessive amount of computational resources [23]. Furthermore, the overset interface alone already postponed flow separation for the isolated wing as shown in Section 4.2, and the influence of the blade tip vortices on the flow close to the wall can get attenuated by the overset interface that lies in between [31]. Hence, the main reason for the discrepancy between flow separation predicted by the RANS simulations and the experiments is not only that the blade tip vortices are weaker in the numerical results but also that the influence of flow gradients close to the wall are predicted incorrectly by the overset interface.

Figure 7.6: Comparison between the time-averaged CFD RANS results and the steady PIV measurement for the velocity component in the x direction over the flap. The pink line connects points of zero axial velocity.

Figure 7.7: Velocity profiles over the flap at three axial coordinates.

Figure 7.9 compares the effect of the propeller on the pressure distribution of the wing profile below the propeller axis between CFD and the experiment. The later separation in CFD not only mitigates the decrease of the suction peak at \(x/c = 0.8\) (compare top right quadrant to top left) but also increases the pressures on the pressure side due to an increased pressure at the trailing edge, whereas the pressure side in the experiment stays mostly unaffected. On the suction side of the leading edge, the propeller has only a small effect on the pressures in both CFD and the experiments as predicted from the isolated propeller analysis in Section 6.2. The surface imperfections of the wing model are evident as kinks in the pressure distribution of all the experimental results, especially close to the leading edge. The increased circulation, and smoother pressure distribution around the leading edge, cause the significantly larger propeller-on lift from CFD than from the experiments. The -on/-off values of the sectional pressure-lift, and pressure-drag coefficients, below the propeller axis, are compared in Table 7.1, where propeller-off means the wing with nacelle and spinner only.
Figure 7.8: Comparison between the instantaneous CFD RANS results and the phase-locked PIV measurement for the out-of-plane vorticity over the flap.

Table 7.1: Sectional pressure-lift and pressure-drag coefficients of the wing profile at $y/c = 0$, for the propeller-off and propeller-on case.

<table>
<thead>
<tr>
<th>Experiment</th>
<th>C$_{l,p}$</th>
<th>C$_{d,p}$</th>
<th>C$_{l,p}$</th>
<th>C$_{d,p}$</th>
<th>ΔC$_{l,p}$</th>
<th>ΔC$_{d,p}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Propeller off</td>
<td>0.097</td>
<td>0.0111</td>
<td>0.101</td>
<td>0.0080</td>
<td>+0.004 (+4 %)</td>
<td>-0.0031 (-28 %)</td>
</tr>
<tr>
<td>Propeller on</td>
<td>0.115</td>
<td>0.0089</td>
<td>0.167</td>
<td>0.0051</td>
<td>+0.052 (+45 %)</td>
<td>-0.0038 (-43 %)</td>
</tr>
<tr>
<td>Off to on</td>
<td>+0.018 (+19 %)</td>
<td>-0.0022 (-20 %)</td>
<td>+0.066 (+65 %)</td>
<td>-0.0029 (-36 %)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

is considered. While the lift coefficient in the propeller-off case compares reasonable well between experimental and numerical results, the drag coefficient decreases by $\Delta C_{d,p} = 0.0031$. This is attributed to higher pressures over the flap in CFD compared to the experiment, as can be seen in the bottom left image of Figure 7.9. With the propeller on, the difference in drag increases further due to the increase in pressures at the trailing edge in the numerical results, contrary to the pressure decrease in the experimental ones. Consequently, the pressure drag decreases more when the propeller is added in the numerical simulations than in the experiments.

In Figure 7.10(a), the thrust of all blades combined, and of a single blade are shown per iteration. The propeller thrust reaches a steady value after about two to three revolutions. Small fluctuations in the thrust coefficient of all blades are also present, but can be attributed to the oscillation in thrust of the individual blades depending on their azimuthal position. The thrust coefficient is calculated from the net thrust, without the influence of the spinner and nacelle. Furthermore, the thrust distribution for the blade closest to the wing, after different numbers of blade revolutions have passed, is shown in Figure 7.10(b). No difference in the thrust distribution between seven and eleven revolutions can be observed. The thrust coefficient of $C_T = 0.34$ is highlighted as red circle in the performance curve of the propeller, obtained from the experiment in Figure 7.11. The numerical result for the propeller thrust compares well to the experimental results, as it is less than 3% lower. This difference can be attributed to a slightly higher velocity field in which it operates due to the weaker flow separation over the flap.
7.1. Comparison to experimental results

Figure 7.9: Pressure distribution of the wing-section at $y/c = 0$, with and without influence of the propeller, from the experiment and CFD. Errorbars have been added to every second measurement point on the suction side of the wing.

Figure 7.10: Convergence plot for the thrust coefficient (a) for all blades combined and for one blade individually, and thrust distributions (b) of the same blade after different amounts of propeller revolutions have passed.
7.2. Influence of the wind tunnel walls

The influence of a closed test section on the measurement results can be split into two parts: two-dimensional and three-dimensional interaction effects. The walls of the test section constrain the air movement inside the test section as the airflow is aligned with them, and the test model inside the wind tunnel represents an obstacle for the surrounding flow. The model reduces the effective cross-section of the test section, which accelerates the flow due to the conservation of mass. Hence, the airflow in the wind tunnel test section is not completely representative of a real-life application, and consequently, wind tunnel measurements need to be corrected, to account for the influence of the wind tunnel walls. Under the assumption of a 2D flow, convenient correction factors can be applied to account for the influence of the walls. Such analytical methods from Barlow et al. [4] will be compared to numerical 2D results.

On the other hand, a boundary layer is formed at the test section walls, which not only further decreases the effective cross-section but also interferes with wall-mounted objects. For a wing covering the entire width of the test section, this wall boundary layer interacts with the boundary layer of the wing. Such a configuration might cause a horseshoe vortex and flow separation at the intersection, hence affecting the lift distribution along the span. In order to identify the influence of the wind tunnel wall boundary layer on the wing’s lift distribution, the 3D flow field needs to be studied.

Both 2D and 3D interaction effects will be analysed in this section. This will be done for the isolated wing first, and then the OTW propeller system. In doing so, a step towards a more real-life environment will be taken. In the previous section, the numerical results inside the confined test section were compared to the experiments. Now, the influence of the constraining walls on the aerodynamic characteristics will be analysed, because only then we have a clean basis for studying the effect of propeller inclination in the next chapter.

7.2.1. Isolated wing

In this section, inviscid and viscous wall effects will be analysed for the isolated wing. The major interference effects which can easily be modelled in 2D are solid model blockage, streamline curvature blockage, and wake blockage. Solid model blockage is due to the finite ratio of the frontal area of the model to the cross-section of the wind tunnel. Typically this ratio is about 0.05 [4]; here it is 0.07. In real-life applications, this ratio would be zero. The blockage causes an increase in the oncoming flow speed, hence increasing dynamic pressure. Furthermore, the wind tunnel walls hinder the curvature of the streamlines of the flow around a body compared to an infinitely large domain. The closer the walls are to the model, the stronger the streamlines are constrained, as the surrounding flow follows the straight walls. In a closed test section, this leads to an increase in the moment coefficient, lift, and angle of attack, because, relative to the straightened flow at the walls, the airfoil seems to have an increased curvature [4]. Similar to the wing, also the low-momentum wake displaces the surrounding flow, which leads to an accelerated flow downstream (outside of the wake), hence an additional backward force component.

To study the effect of model, streamline, and wake blockage, 2D simulations of the isolated wing with and without wind tunnel walls were performed. A visual comparison of the respective pressure distribution in Figure 7.12 reveals a shift towards lower pressures when the walls are present. This is due to a higher
7.2. Influence of the wind tunnel walls

dynamic pressure caused by the model blockage effect. The increased axial velocity over the suction side of the wing is shown in Figure 7.13. The vertical coordinate is normalised with the propeller radius to visualise how blockage affects the axial inflow to the propeller. The velocity profiles were extracted along a vertical line from the wing surface to the top wall of the wind tunnel. For the case without walls, a line with the same length was used. At the top wall of the wind tunnel, the axial velocity is about 5% higher than free-stream and 2.5% higher than without walls. The maximum axial velocity, close to the wing, is only 1.4% higher with walls than without. The increase in velocity close to the wing is less, because the streamlines far away from the model are most displaced. This results in a reduced non-linear velocity gradient of the inflow to the propeller with walls.

![Figure 7.12: Pressure distribution of the 2D airfoil.](image1)

![Figure 7.13: Inflow velocity profile at the chordwise propeller location (propeller not present).](image2)

The sectional lift and drag coefficients of the airfoil with and without wind tunnel walls are shown in Table 7.2. The coefficients are shown for results from 2D and 3D simulations to see how well the 2D coefficients compare to the coefficients in the middle of a 3D wing. Table 7.2 shows that assuming two-dimensional flow in the middle of the wing has only a small effect on the lift and drag coefficients. Therefore, 2D analytical correction factors will be used. The values with wind tunnel walls are the uncorrected values and, ideally, the same decrease in lift and drag can be found when the correction factors from literature are applied. This will be tested by applying the 2D correction methods for a body spanning the tunnel from Ref. [4] to the 2D numerical results with walls. The correction factors used for solid body blockage $\epsilon_{sb}$, and wake blockage $\epsilon_{wb}$ are:

$$\epsilon_{sb} = 0.74 \frac{W}{A^{3/2}} = 0.0298$$  \hspace{1cm} (7.3)

$$\epsilon_{wb} = \frac{c}{4 \cdot h} C_{d,u} = 0.0045$$  \hspace{1cm} (7.4)

where $W$ is the volume of the wing model, $A$ is the cross-section of the test section, $c$ the chord, $h$ the height of the tunnel from above to below the wing, and $C_{d,u}$ is the uncorrected drag coefficient. The stream line curvature blockage is expressed by a factor $\sigma$:

$$\sigma = \frac{\pi^2}{48} \left(\frac{c}{h}\right)^2 = 0.0689$$  \hspace{1cm} (7.5)

With the factors $\epsilon_{sb}$ and $\epsilon_{wb}$ combined as $\epsilon = \epsilon_{sb} + \epsilon_{wb}$, the corrected velocity, dynamic pressure, Reynolds number, lift, and drag can be obtained from:

$$V = V_u \cdot (1 + \epsilon)$$  \hspace{1cm} (7.6)

$$q = q_u \cdot (1 + 2\epsilon)$$  \hspace{1cm} (7.7)

$$Re = Re_u \cdot (1 + \epsilon)$$  \hspace{1cm} (7.8)

$$C_l = C_{l,u} \cdot (1 - \sigma - 2\epsilon)$$  \hspace{1cm} (7.9)

$$C_d = C_{d,u} \cdot (1 - 3\epsilon_{sb} - \epsilon_{wb})$$  \hspace{1cm} (7.10)
The corrected lift and drag coefficients from Equations (7.9) and (7.10), decrease by 13% and 9%, respectively, to $C_l = 0.086$ and $C_d = 0.0132$. Table 7.2 shows that these values compare poorly, especially the lift coefficient, to the 2D numerical solutions without walls. In general, empirical relations like the wall corrections may differ from numerical results because often they are based on observations, shape factors, or measured variables like $\epsilon_{wb}$ is. The relatively large discrepancy, however, can be explained by an overestimation of the factors for the solid body blockage and stream line curvature. The model blockage merely increases the dynamic pressure around the wing. As the pressure distribution in Figure 7.12 shows, this increases suction on both sides of the wing due to its almost symmetrical profile. Hence, it is not very effective in increasing lift. Also the streamlines are likely less affected than estimated since their curvature decreases quickly towards the walls, due to the relatively low lift of the wing. However, the velocity increase - and with this, also the dynamic pressure and Reynold's number - is fairly accurately accounted for at the wind tunnel wall, where an increase of 3% (compared to the 2.5% from CFD) is calculated by Equation (7.6).

Table 7.2: Sectional lift and drag coefficients from the 3D and 2D results of the isolated wing, with and without wind tunnel walls.

<table>
<thead>
<tr>
<th></th>
<th>With walls (WT)</th>
<th>Without walls (FF)</th>
<th>WT to FF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$C_l$</td>
<td>$C_d$</td>
<td>$C_l$</td>
</tr>
<tr>
<td>From 3D</td>
<td>0.100</td>
<td>0.0148</td>
<td>0.093</td>
</tr>
<tr>
<td>From 2D</td>
<td>0.099</td>
<td>0.0145</td>
<td>0.094</td>
</tr>
<tr>
<td>3D to 2D</td>
<td>-0.001 (-1 %)</td>
<td>-0.0003 (-2 %)</td>
<td>+0.001 (+1 %)</td>
</tr>
</tbody>
</table>

Three-dimensional corrections are not as straightforward anymore since the boundary layers of the wing and wall interact with each other. This can cause a vortex inside the intersection area, called horseshoe vortex. Since there are already large discrepancies for the 2D corrections, no attempt will be made to compare the 3D numerical results with 3D analytical correction methods. Hence, the influence of the walls on the 3D interaction at the wing-wall junction will only be studied numerically by changing the boundary condition of the wind tunnel wall intersecting with the wing model from no-slip to symmetry, hence removing the boundary-layer. The resulting spanwise lift distribution, obtained when a boundary-layer is only present on the wing, but not on the sidewall is shown in Figure 7.14, labelled as Symmetry wall. The fact that this lift distribution is completely horizontal and only marginally lower than the 2D lift coefficient of $C_l = 0.099$, shows that 2D assumptions to the lift coefficient can be made if there is no interaction with a boundary-layer on the sidewall of the wind tunnel. When the sidewall boundary condition is changed to a no-slip wall, the mesh close to the wall needs refinement to correctly simulate the boundary-layer. Therefore, the lift distributions for two different levels of wall refinement are compared here. For a wall distance of $10 < y^+ < 80$ a wall function approach is used by the solver. In case of a finer mesh of $0.05 < y^+ < 0.6$ the Navier-Stokes equations are solved up to the wall.

![Figure 7.14: Lift distribution for different wall refinement.](image)

An explanation for why the lift distributions differ between the two levels of refinement can be found by inspecting the junction flow in Figure 7.15, with wall functions (left) and without (right). For both cases, the streamlines close to the sidewall are displaced by a horseshoe vortex around the leading edge of the wing.
The incoming, thick, boundary layer of the wind tunnel wall collides with the boundary layer of the wing in front of the leading edge and separates from the wall locally, facing an adverse pressure gradient of the stagnant flow in front of the leading edge. Then, the separated flow forms a vortex band along the wing chord in the wing-wall junction, which transports high-momentum flow from outside the sidewall’s boundary layer into the junction area. In Figure 7.15 this is visible in the cut-ins of increased total pressure in the corner between the wing and wind tunnel wall. Up to the flap, no major differences can be observed between the two levels of wall refinement. Over the flap, however, a large flow separation area exists only for the finer sidewall mesh. This corner flow separation is likely to be induced by the interaction of the boundary layers. Gand et al. [28] show that the intensity of such a corner flow separation is influenced by the thickness of the incoming boundary layer. This would explain why the separation behaviour over the flap is different between the two wall refinements, as the boundary layer thickness of the wind tunnel wall may change depending on the mesh refinement and calculation approach.

As a result, the spanwise lift distribution differs between the finer and coarser wall refinement, as seen in Figure 7.14. The corner flow separation leads to a strong lift reduction close to the sidewall. On the other hand, the lift distribution for $10 < y^+ < 80$ exhibits increased lift at the wall, which may be a form of vortex lift due to the horseshoe vortex. To save computational time, these simulations were done for a half-wing model. Hence, in the middle ($y/b = 0$) is a symmetry boundary condition which mirrors the effect of the sidewall such that even in the middle an influence of the sidewall-interaction on the lift can be found, which will also be the case for the full model. However, within the extent of the propeller radius, the lift distributions are horizontal. This suggests that although the inflow velocity into the propeller might differ in an absolute sense based on the wall refinement, no additional spanwise velocity gradient should be expected.

For the following OTW simulations, the wall-functions approach is used, because the calculations with inflation layer on the sidewalls took four times longer and the lift coefficient of $C_L = 0.1$ in the middle of the wing is closer to the experimental results. The 3D lift coefficient of the wing, obtained without a boundary-layer at the sidewall is 6% ($\Delta C_L = -0.006$) lower than the 3D lift coefficient with a boundary-layer using wall-functions at the sidewall. As an estimation of the combined effects of blockage and boundary-layer-interaction, their respective changes in lift of $\Delta C_L = -0.007$ and $\Delta C_L = -0.006$ can be added up, which results in an expected lift decrease of $\Delta C_L = -0.013$ when the walls are removed, under the assumption that the propeller has a negligible effect on the influence of the wind tunnel walls.

![Figure 7.15: Visualisation of the junction flow at the intersection between the wing and wind tunnel wall. The solid black lines follow the streamlines close to the wall. Contours of the skin friction coefficient $C_f$ are shown on the sidewall and wing, and total pressure contours ($C_p,t$) are shown for x-planes in the junction. Vortex structures are visualised by an iso-surface of the Q-Criterion at $Q = 1000 \text{ s}^{-1}$.](image_url)
7.2.2. Wing-propeller system

In this section, the results of the integrated wing-propeller system obtained without wind tunnel walls will be compared to the ones obtained with wind tunnel walls, which were discussed previously in Section 7.1. The convergence of the lift and drag of the wing, and the thrust of all blades and an individual blade, are shown in Figure 7.16. Again, averaged over the last two propeller rotations, the lift and drag coefficients are:

\[
C_L = 0.126 \quad (7.11)
\]
\[
C_D = 0.0135 \quad (7.12)
\]

Compared to the simulations with wind tunnel walls (Equations (7.1) and (7.2)), this represents a decrease in lift and drag by \( \Delta C_L = -0.017 \) and \( \Delta C_D = -0.0008 \), respectively. Hence, the lift decrease differs only slightly to the estimation from the isolated wing. A reason for the difference may be that adding the propeller changes the relative importance of the individual blockage effects in a non-linear way. For instance, the streamline curvature effect may become more dominant due to the increased circulation of the wing, caused by the propeller. The solid model and wake blockage, however, may decrease due to the locally accelerated flow in the propeller slipstream causing a decrease in free-stream velocity outside of it, when the walls are present. This is known as slipstream blockage [4]. Moreover, the 2D simulations were done without the spinner and nacelle which also contribute to the solid model blockage once the propeller is added.

Figure 7.16: Convergence plot for the lift, drag, thrust of all blades, and thrust of blade 1 (baseline configuration without wind tunnel walls).

The averaged thrust of \( C_T = 0.341 \) without walls (Fig. 7.16) is identical to the averaged thrust with walls of \( C_T = 0.341 \). Similarly, the thrust distribution of the propeller disc is the same with wind tunnel walls and without them. The trust distribution is shown in the Appendix in Figure D.1. Hence, it can be assumed that the wind tunnel walls have no significant influence on the propeller thrust. This is expected because the walls are five to six propeller radii away from the propeller. Furthermore, two effects caused by the walls which were identified in Section 7.2.1 might cancel each other out: the increased axial velocity into the propeller and the reduced non-linear inflow gradient. The non-uniform inflow is increased but its non-uniformity is decreased.

The spanwise lift and drag distributions of both configurations are compared in Figure 7.17. Here, the distributions are obtained by phase-averaging the transient results over 30 time steps, the equivalent of 60 degrees of propeller rotation in total. For conciseness, the abbreviations WT and FF, standing for wind tunnel and far-field, are used to label the results with wind tunnel walls (WT) and without (FF), respectively. The lift with wind tunnel walls is higher over the entire span, due to the blockage effects. At the intersection of the wing and the sidewalls, an additional lift increase can be found, which is attributed to the horseshoe vortex.
7.2. Influence of the wind tunnel walls

At the wing-wall junction a local decrease and increase in drag can be found. This is caused by a combination of reduced pressure drag due increased leading edge suction increased friction drag due to the vortex washing over the wing. The spanwise drag distribution reveals a larger drag reduction at the propeller location when the wind tunnel walls are present. Breaking down the drag into friction and pressure drag shows that this local drag reduction is due to a decrease in pressure drag, which is typical for OTW applications. However, this drag reduction is usually due to increased leading edge suction, i.e. induced thrust \([34, 47]\). Hence, it should be visible as an increased suction peak at the leading edge. Figure 7.18 shows that this is not the case. On the contrary, the pressure around the leading edge on the suction side is very similar with and without walls. Only on the lower wing surface, a difference is visible as the pressure is slightly increased with wind tunnel walls, due to an increased circulation when the walls are present. To find out whether the difference in pressure drag between walls and no walls is caused at the leading edge or trailing edge, the pressure coefficient in the \(x\)-direction is shown in Figure 7.19 along the vertical coordinate of the wing profile non-dimensionalised with the profile thickness. A positive \(C_{p,x}\) indicates pressure directed in the free-stream direction, hence drag. Figure 7.19 shows that the main cause for the decreased sectional pressure drag with walls is a decrease in \(C_{p,x}\) over the entire surface of the suction side of the flap when the walls are present. This decrease in pressure is caused by the slipstream blockage decreasing the velocities between the slipstream and flap, compared to the case without walls. The slipstream blockage effect is visualised in the velocity profile in Figure 7.20, extracted along the black line in Figure 7.21.

![Figure 7.17: Spanwise lift and drag coefficients \(C_l\) and \(C_d\) averaged over 60 degrees of propeller rotation for the cases with and without wind tunnel walls.](image)

Figure 7.17: Spanwise lift and drag coefficients \(C_l\) and \(C_d\) averaged over 60 degrees of propeller rotation for the cases with and without wind tunnel walls.

![Figure 7.18: Pressure distribution of the wing profile at \(y/b = 0\) averaged over 60 degrees of propeller rotation for the cases with and without wind tunnel walls.](image)

Figure 7.18: Pressure distribution of the wing profile at \(y/b = 0\) averaged over 60 degrees of propeller rotation for the cases with and without wind tunnel walls.

![Figure 7.19: Axial component of the pressure coefficient for the wing profile at \(y/c = 0\).](image)

Figure 7.19: Axial component of the pressure coefficient for the wing profile at \(y/c = 0\).
7. Baseline over-the-wing configuration

In Table 7.3, the sectional lift and drag coefficients with the propeller off and on are shown for the case with, and without wind tunnel walls. The propeller-off coefficients are obtained from simulations of the wing with the nacelle and spinner only, without propeller blades. This enables the isolation of the effect of the propeller thrust on the wing by calculating the change in lift and drag coefficients from the propeller off and propeller on coefficients. It can be inferred from Table 7.3 that the propeller-on drag of the wing profile below the propeller axis increases, while the drag in the propeller-off case decreases, when the walls are removed. This is attributed to the lack of streamline blockage without the walls. Consequently, the propeller reduces the drag coefficient more when the wind tunnel walls are present. The propeller-induced increase in lift coefficient is practically unaffected by the removal of the walls, since the change to far-field boundary conditions decreases the propeller-off and propeller-on lift almost by the same amount.

Table 7.3: Sectional lift and drag coefficients of the wing profile at \( y/c = 0 \), for the propeller-off and propeller-on case, with and without walls.

<table>
<thead>
<tr>
<th></th>
<th>With walls (WT)</th>
<th>Without walls (FF)</th>
<th>WT to FF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>( C_L )</td>
<td>( C_D )</td>
<td>( \Delta C_L )</td>
</tr>
<tr>
<td>Propeller off</td>
<td>0.099</td>
<td>0.0161</td>
<td>-0.009 (-9 %)</td>
</tr>
<tr>
<td>Propeller on</td>
<td>0.160</td>
<td>0.0123</td>
<td>+0.01 (-6 %)</td>
</tr>
<tr>
<td>Off to on</td>
<td>+0.061 (+62 %)</td>
<td>-0.0038 (-24 %)</td>
<td>+0.06 (+67 %)</td>
</tr>
</tbody>
</table>

7.3. Aerodynamic interaction phenomena

Since the influence of the wind tunnel walls on the aerodynamics of the wing and propeller has been analysed in the previous section, the remainder of this thesis will focus on the configuration without walls. In this section, a closer look will be given into the characteristic aerodynamic interaction effects between the wing and propeller. In particular, the discussion will be focused on the velocities and pressures induced by the propeller and their influence on flow separation over the flap.

7.3.1. Propeller-induced pressure and velocity fields

The downwash of the wing induces a vertical velocity component in the slipstream which causes a downward deflection of the stream tube behind the propeller. However, Figure 7.22(a) and (b) show that the propeller's slipstream does not completely follow the flap deflection, because the change in vertical momentum over the flap is not large enough to re-direct the propeller slipstream. This results in an area of low axial velocity over the flap, similar to the flow in a divergent channel. The induced pressures and velocities in Figure 7.22(b) are calculated as \( \Delta C_p = C_{p}^{on} - C_{p}^{off} \), where the propeller-off case is with the nacelle and dummy-spinner. The local decrease in \( V_x \) is evident in the induced-velocity vectors in the negative axial direction in the range...
of $0.8 < x/c < 0.9$, in Figure 7.22(b). Consequently, the wing’s suction peak behind the propeller location decreases.

The velocities induced by the propeller are largest in the outboard part of the slipstream, where the propeller loading is the largest. Ahead of the propeller, the induced velocity vectors vanish quickly, indicating a relatively weak upstream effect. At the lower edge of the propeller diameter, the wing shields the streamtube which leads to less radially induced velocities, hence a more axially aligned inflow, contrary to the upper propeller radius. The induced velocities parallel to the wing’s surface further intensify the suction ahead of the propeller and on the wing. The pressure side of the wing is fairly unaffected by the propeller and only reacts to an increase in circulation or change in stagnation point.

Figure 7.22: Phase averaged axial velocity and streamlines in the mid-plane at $y/c = 0$ (a). Phase averaged induced pressure and velocities in the mid-plane at $y/c = 0$ (b).

An instantaneous close-up view of the spanwise vorticity, defined as $C_{\omega y} = \omega_y/(2 \Omega)$, over the flap region in Figure 7.23 reveals the blade tip vortices. Here, the vectors again display the propeller-induced velocities, but in this case instantaneous, instead of phase averaged. Below the second vortex ($x/c \approx 0.84$), the strongest reduction in momentum induced by the propeller can be found. This is not only due to the divergence of the slipstream from the flap, but also due to the additional induced velocities by the blade tip vortices directed in the negative axial direction. The velocity profiles over the flap in Figure 7.24 confirm that the propeller reverses the flow direction close to the wall, and reduces the momentum at a distance between $d/c = 0.01$ and $d/c = 0.025$ to the wall. The profiles were extracted along the black survey line in Figure 7.23. Following the work by Murray et al. [51], a horizontal slice, parallel to the wing’s upper flat surface, was cut in the middle of the tip gap at $z/c = 0.0564$, indicated by the dashed line in Figure 7.23. No reversed flow was found in the middle of the tip gap, similar to the findings in Ref. [51] for a comparable advance ratio ($J = 1.05$), despite differences in the ratio of the boundary layer height to propeller diameter. Nevertheless, plotting the vorticity contour for the slice in the tip gap as in Figure 7.25, visualises the downward deflection of the slipstream and different stages in the diffusion process of the blade tip vortices. The green contour-line indicates the blade position by projecting the airfoil shape of the propeller blade at the hub onto the plane.

The propeller-induced pressures in the flow field in Figure 7.22(b) are reflected on the wing’s suction side as decreased static pressure ahead of the propeller location and increased pressure behind it. Figure 7.26 shows that this is mostly a local effect as it is most prominent from $x/c = 0.5$ onwards and $y/c = \pm 0.15$ laterally (the propeller radius is equal to $y/c = 0.0975$). The pressure distribution of the wing profile at $y/c = 0$ (Fig. 7.27), however, also shows a small increase in suction at the leading edge as well, due to a small increase in dynamic pressure. The propeller reduces the suction peak of the flap hinge and shifts it more towards the leading edge. Interestingly, there is an area of reduced pressure on the flap even behind the propeller ($0.85 < x/c < 0.95$) followed by an increase in pressure ($0.95 < x/c < 1$). The deltas in this range are relatively small, which makes it difficult to justify whether they are caused by numerical errors, for instance due to the
averaging of the unsteady pressures, or an actual flow phenomena. An explanation could be that due to an upstream movement of the stagnation point on the flap, more high-pressure from the pressure side of the wing reaches the suction side. Hence, a change in the circulation of the airfoil which is also indicated by slightly higher pressures on the pressure side of the leading edge when the propeller is on (Fig. 7.27).

The transient response of the pressure distribution to the blade position is shown in Figure 7.28. The largest suction-gain can be observed at a phase angle of $\phi = 60^\circ$, or multiples of it, which are the angles at which a blade is closest to the wing. At these blade positions, the low pressures of the blade’s suction side merge with the low pressures of the wing’s suction side, increasing the suction peak. On the other hand, the pressure side of the blade also increases the pressure on the wing behind it. Consequently, the backward facing component of the suction peak is reduced due to the propeller’s position just in front of the curved part of the flap. As the propeller blade moves away from the surface, the suction augmentation fades and the peak shifts more toward the trailing edge. Since the flap-surface is the only wall of the wing with a component of its normal vector pointing opposite to the theoretical flight direction, it is evident that this transient shifting of the suction peak, and the fluctuations of the pressures on the flap, influence the drag of the wing profile.

Figure 7.29 confirms that the variations of the suction peak influence the sectional drag. Specifically, the pressure drag is the lowest at the phase angles where a blade is closest to the wing, which is a direct result of the decreased backward-facing component of the suction peak on the flap. At an angle of $\phi = 50^\circ$ the suction
peak is significantly weaker than at $\phi = 60^\circ$, however, along the flap's suction side, the pressure is still slightly lower, which is enough to increase the drag marginally. Chordwise positions below $x/c \approx 0.7$, as well as the pressure side of the wing, are not affected by the phase angle, as can be inferred from Figure 7.28. Due to the profile of the wing consisting of two flat plates over a large range of the chord, the pressure drag is actually consistently lower than the friction drag. The friction drag is not affected by the propeller. Since the changes in both lift and drag are highly sensitive to flow separation over the flap, the following section describes this phenomena in more detail.
7.3.2. Flow separation on the wing

In this section, a closer look will be given at how the induced pressures and velocities influence the flow over the flap. As an overview, the skin friction coefficient of the suction side of the wing is given in Figure 7.30. The view is limited to the area of influence of the propeller, which does not exceed $y/c = \pm 0.2$ spanwise and about $x/c = 0.5$ ahead of the propeller. This was found by plotting the skin friction difference between the propeller-on, and propeller-off case (Fig. E.1 in the Appendix). The shear-lines in Figure 7.30 show that in front of the propeller, close to the wall, the flow is mostly two-dimensional. Only closer to the propeller location, the contraction of the streamtube into the propeller deflects the shear-lines. Behind the propeller, the flow close to the flap-surface follows the curvature of the flap, gets deflected towards the middle of the wing, and then moves upstream as reversed flow, despite the influence of the overset interface previously shown in Figure 4.14. This is attributed to the mass flow deficit in the region between the flap wall and propeller-slipstream diverging from it. The surface of the wing’s suction side limits the radial velocities at the lower blade tips, hence constraining the contraction of the slipstream, which can lead to local flow reversal on a wall [51]. As a comparison, reversed flow being sucked into the propeller location from downstream and lateral was found by Murray et al. [51] only for a higher thrust, and a lower advance ratio. However, the authors only investigated the flow in a plane away from a flat wall, and not directly at the wall. Moreover, the propeller location in Ref. [51] was not followed by a deflected flap which thickens the boundary layer due to its adverse pressure gradient. Interestingly, a postponement of flow separation over a large width in the spanwise direction that extends the influence of the overset interface is present in Figure 7.30. This can be inferred by comparing...
the chordwise position of the line of converging shear-lines to the dashed line indicating the location of flow separation for the isolated wing. This is attributed to an increased momentum in the boundary layer in front of the propeller, because of the favourable pressure gradient over the flat plate part of the wing. Hence, the boundary layer becomes thinner and more likely to withstand the adverse pressure gradient of the flap. The boundary-layer profiles 0.5\(R\) ahead of the propeller, at three spanwise locations, in Figure 7.31 confirm an increase in momentum close to the wall, even below the outboard parts of the slipstream at \(y/c = \pm 0.075\). Consequently, only below the axis of the propeller, where the slipstream is closest to the wing, the mass flow deficit is strong enough to trigger flow separation in the axial direction.

![Figure 7.30: Phase averaged skin friction coefficient on the wing’s suction side and wall shear-lines. The dashed line indicates the chordwise location of separation for the isolated wing. The colourbar does not span the full range of friction coefficient values to improve readability.](image1)

![Figure 7.31: Boundary-layer profiles 0.5\(R\) ahead of the propeller at three different spanwise locations.](image2)

The components of the locally reversed velocity in axial and spanwise direction are relatively low as can be inferred from Figure 7.32. The fluid domain is cut vertically in the middle of the wing, and at the spanwise locations where approximately the largest delay of flow separation was observed. In the middle plane, flow in the negative x-direction can only be found very close to the wall. In the outer two planes, axial velocities close to zero can only be found at the trailing edge. On the other hand, the cross-flow components of the velocity are larger in these planes than in the middle as the flow gets sucked in from both sides towards the middle plane.
To study the time-dependent behaviour of the skin friction of the middle wing profile, its coefficient is shown in Figure 7.33(a) for different phase angles. Depending on the blade position, various small spikes appear at the axial position of the propeller ($x/c = 0.79$) as well as low amplitude oscillations over the flap. However, no significant change in the location of lowest skin friction due to the movement of the blade can be identified. The skin friction becomes zero when flow reversal occurs. Hence, it can be concluded that the point of flow separation has a temporal dependency on the position of the blade tip vortices due to their momentarily increase in adverse pressure gradient but the chordwise extent of their influence is small. On average, flow is separating with the propeller more upstream than without it, i.e. for the isolated wing (see Fig. 7.33(b)). However, the separation point is still behind the propeller location, thus behind the flap hinge. Hence, instead of directly forcing flow separation, the adverse pressure gradient at the propeller location only weakens the boundary layer such that its momentum is not sufficient anymore to withstand the additional adverse pressure gradient of the flap, at least below the centerline of the propeller. This hypothesis is in line with findings of an experimental study of the same propeller and thrust setting over a flat plate, without a flap, which showed no flow reversal at the wall, despite an even smaller tip clearance [19].

Figure 7.33: Unsteady and averaged skin friction coefficient on the suction side of the wing profile at $y/c = 0$. In Figure (b) the averaged skin friction coefficient (Avg) is compared to the skin friction of the isolated wing without the propeller (Iso).
7.4. Propeller loading distributions

The propeller thrust averaged over the last two full rotations is $C_T = 0.341$, as visible in the convergence plot in Figure 7.16. Compared to the isolated thrust of $C_T = 0.336$ this represents a thrust increase of $\Delta C_T = 0.005$, which is less than 2 % of the isolated thrust. However, this is very close to the uncertainty due to the discretisation error established in Section 4.1.2 which makes it questionable whether the propeller thrust really increased. Yet, an increase in propeller thrust, or at least a negligible change, is also reported in literature by Cooper et al. [16], and Johnson and White [34]. Although the non-uniform inflow due to the presence of the wing (Fig. 6.3(a)) suggests a decrease in propeller thrust, a small increase, or at least compensated decrease, can be explained by considering the postponed flow separation on the flap, behind the outboard parts of the propeller area (Fig. 7.30). If the flow stays attached to the flap longer, the streamlines around the wing are deflected more downwards, which results in a larger effective angle to the propeller area. Hence, the tangential velocity component increases, and the blade angle of attack increase for the up-going blades become more dominant than the decrease due to the high inflow velocities over the wing. At the down-going blade, however, the velocities induced by the wing decrease the local tangential velocities as shown in Figure 7.34. The velocities shown in Figure 7.34 are calculated as wing on minus wing off, hence, the results of the isolated propeller are subtracted from the results of the combined system. It can be seen that the wing induces not only a velocity increase in the axial direction but also in the tangential direction of the blade. This decreases the local thrust due to an increase in local advance ratio, and compensates the thrust increase of the up-going blades such that the effect on the total thrust is small. The resulting thrust distribution and its change relative to the isolated propeller are shown in Figure 7.35(a) and (b), respectively.

![Figure 7.34: Velocities induced by the wing, in front of the down-going blade.](image)

A similar asymmetry in the azimuthal thrust distribution, due to an increased velocity component in the negative $z$-direction, was also predicted with a BEM code, and confirmed by experimental measurements of the total pressure a wake plane, by Marcus et al. [40], and is shown in Figure 7.36. Since the propeller was spinning in the opposite direction than in the current study, the increase can be found on the other side of the disc plane, however, still in the half-disc of the up-going blade. The propeller in Ref. [40] was close to the trailing edge of a wing with a lift coefficient of $C_L = 0.5$, without flap deflection, and at an advance ratio of $J = 0.7$. For the isolated propeller in Ref. [40], the advance ratio of $J = 0.7$ resulted in a thrust coefficient of $C_T = 0.12$.

The prediction of the thrust distribution based on the flow field around the isolated wing (Fig. 6.3(a)) turns out to be invalid because the propeller changes the pressure distribution of the wing. Hence, the velocities induced by the wing change, and with this, the inflow to the propeller changes. This highlights how deeply connected the aerodynamic two-way-interaction between the wing and propeller is. The reason why OTW propellers often experience a decrease in thrust (e.g. in Ref. [40, 47]) is because the super-velocities close to the wing significantly reduce the blade angle of attack in the lower areas of the propeller disc. However, relevant for the significance of this effect is the chordwise position of the propeller. In the experiments...
by Marcus et al. [40] only a more forward position of the propeller lead to a thrust reduction in the lower propeller disc, since the vertical velocity gradient over the wing is stronger closer to the leading edge.

![Thrust and Change in Thrust](image)

**Figure 7.35:** Thrust distribution on the propeller disc (a) and difference with respect to the isolated propeller (b). The range of the colourmap was defined for better comparability to the inclined propeller thrust distributions in Section 8.2.

The difference in blade thrust depending on phase angle is also visible in the thrust distributions in Figure 7.37. A blade at \( \phi = 90^\circ \) produces more thrust than a blade at \( \phi = 270^\circ \). Thrust distributions of blades at \( \phi = 0^\circ \) and \( \phi = 180^\circ \) deviate only moderately from the isolated blade, because the angle of attack effect is the weakest at these positions. The lower blade at \( \phi = 0^\circ \) creates less thrust than the top blade, and also less than the isolated blade, which is a result of the increased inflow velocities.

![Total Pressure Distribution](image)

**Figure 7.36:** Total pressure distribution in the disc plane. Image adapted from Ref. [40].

![Thrust Distribution](image)

**Figure 7.37:** Thrust distribution for different phase angles compared to the isolated blade.

Not only the thrust but also the torque of \( C_Q = 0.1022 \) of the OTW propeller is similar to the isolated propeller \( (C_Q = 0.1023) \). Since the simulations were performed at a constant angular velocity, also the shaft power coefficient of \( C_P = 0.642 \), defined as \( C_P = P_s/(\rho n^3 D_p^5) \), stayed practically unaffected with respect to the isolated power of \( C_P = 0.643 \) (decrease in \( P_s \) of 0.1%).
7.5. Wing loads

The main figures of merit that will be considered regarding the forces on the wing, are the lift and drag coefficients, their respective deltas, and the resulting lift-to-drag ratio. First, these coefficients will be calculated for the wing as isolated system, influenced by the propeller. Then, to represent the integrated OTW-system as a whole, a corrected lift-to-drag ratio will be given to account for the change in propeller trust affecting the drag of the combined propeller-wing system.

The averaged spanwise lift, drag, and lift-to-drag ratio of the wing influenced by the propeller are shown in Figure 7.38 as "Propeller on". As a reference, the same figures of merit are given for the isolated wing, and the wing only influenced by the nacelle and bladeless spinner ("Propeller off"). The locally induced suction by the propeller results in a peak of the lift in the middle of the wing, below the propeller axis. However, the lift enhancement extends over the propeller diameter such that even at the lateral ends of the wing a larger lift can be found, compared to the isolated wing. This might indicate that the symmetry-walls at the sides are still too close to the propeller axis, despite being more than five times the radius away. As a comparison, in the work by Müller et al. [47] a difference in lift of similar order of magnitude can be found at the side walls as well. However, since here the absolute lift values are significantly lower, the same difference in lift has relatively a larger contribution. In Ref. [47], the distance of the side walls to the propeller axis was less than four times the propeller radius. Contrary to the centred lift peak, the lowest drag is shifted below the outboard part of the up-going blade because of the higher pressures on the flap here, caused by the increased thrust for the up-going blades. As a result, the peak in the lift-to-drag ratio is also shifted in the same direction.

Figure 7.38: Spanwise lift, drag, and lift-to-drag ratio of the wing, averaged over 60 degrees of propeller rotation.

The unsteady fluctuations of the lift and drag depending on the propeller rotation are shown in Figure 7.39. The lift peak follows the counter-clockwise rotation of a propeller blade and diminishes as the blade moves away from the wing. At a phase angle of $\phi = 30^\circ$, it can be observed that a new lift peak starts to develop left of the propeller axis at $y/b = 0.05$, following the incoming blade. The fact that the lift peak due to the departing blade ($y/b = -0.05$) at $\phi = 30^\circ$ is still larger than the one at $y/b = 0.05$ shows that the up-going blade momentarily has a stronger influence on the wing’s lift than the down-going blade. This is in line with the thrust distribution on the propeller disc in Figure 7.35. The up-going blade creates more thrust than the down-going one. Hence, there is an asymmetry in propeller-induced velocities, and suction, in front of it, which increases the lift below the up-going blade. On average, however, the lift is still centred in the middle of the wing. The asymmetry in thrust also results in an increased pressure downstream of the up-going blades than the down going blades. Hence, the suction on the flap, and with this the drag, is decreased more in the range $-0.1 < y/b < 0$ than $0 < y/b < 0.1$.

For a distributed propulsion application it is not ideal to have wide gaps between adjacent propulsors. Typically, the propellers are as close to each other as propeller performance penalties allow, to increase the wing loading. Therefore, it makes more sense to calculate the deltas in lift and drag over a finite wing segment below the propeller than over the whole span of the isolated wing model. To obtain the 3D increments in lift
and drag over the finite wing segment, the difference between the spanwise, propeller-on, lift and drag forces and the propeller-off forces are integrated over the span of the segment. The resulting $\Delta L$ and $\Delta D$ forces are then divided by the chord $c$, the span of the wing segment $b^\prime$, and the dynamic pressure of the free-stream $q_\infty$ to obtain the force-coefficients. Figure 7.40 shows the trend in lift increase and drag decrease with increasing span of the wing segment. Naturally, the lift enhancement and drag reduction decrease with increasing span, since the relative influence of the propeller on the wing segment fades. Wingspans below a ratio of $b^\prime/D = 1$ are shown, however, the ratio of 1 is a lower feasibility bound to the wingspan as propellers cannot be placed closer than one diameter to each other without staggering them in chordwise direction.

These deltas can then be added to the respective lift and drag coefficients of the isolated wing segment to obtain the augmented lift-to-drag ratio. In order to account for the change in thrust due to the integration effects, thus allow for comparability at different thrust levels, the lift-to-drag ratio will be corrected with the thrust delta $\hat{T}_c = \hat{T}_c - \hat{T}_c^{iso}$ similar to the procedure by Müller et al. [47]:

$$\frac{L^*}{D} = \frac{C_{iso}^L + \Delta C_L}{C_{iso}^D + \Delta C_D - \Delta \hat{T}_c}$$

(7.13)

The lift and drag coefficients $C_{iso}^L$ and $C_{iso}^D$ are obtained from the isolated wing segment of span $b^\prime$. The thrust coefficient $\hat{T}_c$ is not to be confused with $T_c$, since $\hat{T}_c$ is relative to the reference area of the wing segment ($\hat{T}_c = T/(c \cdot b^\prime \cdot q_\infty)$) instead of the propeller area. For a wing segment of span $b^\prime/D = 1.1$, the corrected lift-to-drag ratio can be calculated as:

$$\frac{L^*}{D} = \frac{0.093 + 0.050}{0.0137 - 0.0018 - 0.002} = 14$$

(7.14)

It should be noted that due to the positive thrust-delta, the corrected lift-to-drag ratio is larger than the traditional lift-to-drag ratio of the segment would be at the same lift coefficient.
Inclined over-the-wing configuration

Now that the baseline aerodynamic interaction effects are properly understood, this chapter focuses on how these effects change when the propeller is inclined. For this, first the characteristic aerodynamic interaction effects of the inclined configuration, without wind tunnel walls, will be analysed, and compared to the baseline case. Then the resulting effects on the propeller thrust and wing loads will be evaluated. Finally, the overall propulsive efficiency of the two systems will be compared for a constant shaft power. The simulation of the inclined OTW propeller system was run until the same change in lift, and reduction of the residuals, per time-step was reached as for the baseline configuration. Based on these convergence criteria, the simulation was stopped after seven unsteady propeller revolutions. Phase averaged results are again based on the average of 30 time-steps of two degrees of rotation. A comparison of the convergence of lift, drag, and thrust, to the baseline configuration as in Figure 8.1, already reveals a decrease in lift and stronger thrust fluctuations of the inclined propeller blades. In the following sections, the reason for these differences, and possible similarities, will be explained.

Figure 8.1: Unsteady convergence plot for the lift, drag, thrust of all blades, and thrust of blade 1 (inclined and baseline configuration without wind tunnel walls).
8.1. Aerodynamic interaction phenomena
The analysis of the characteristic aerodynamic interaction effects will be focused on the propeller-induced pressures and velocities, and the flow separation over the flap. In the following, these topics will be discussed and the findings will be compared to the baseline configuration.

8.1.1. Propeller-induced pressure and velocity fields
When the propeller is deflected with the flap, not only the downwash of the wing introduces a vertical velocity component in the slipstream, but also the inclination of the propeller directs the slipstream more downward, as seen in Figure 8.2(a). The slipstream still diverges from the flap surface but less than in the baseline configuration. Higher axial velocities can now be found in the top half of the slipstream, contrary to the lower-half location for the baseline configuration. Below the nacelle, the velocity in the slipstream is lower than above it because of less induced velocities by the propeller, as seen in Figure 8.2(b). This is in line with the prediction of reduced propeller thrust close to the wing, due to the non-uniform inflow in Figure 6.3 from Section 6.1. The reduced induced velocities, resulting from the reduced thrust compared to the baseline propeller thrust, decrease the dynamic pressure over the wing compared to the baseline wing. Combined with the smaller static pressure reduction by the propeller ($\Delta C_p$), this explains the reduced lift of the wing compared to the baseline configuration. The hypothesis of a locally reduced propeller thrust compared to the baseline case will be confirmed in Section 8.2. Similar to the baseline case, negative axial velocity components are induced directly behind the curvature of the flap, due to the hindering of slipstream contraction by the flap surface. Furthermore, the flat part of the wing's suction side is not aligned with the propeller axis anymore, which leads to a less aligned inflow to the propeller and a larger radial component of the induced velocities at the lower blade tips.

The blade tip vortices are closer to the wall in the inclined configuration than in the baseline case, therefore, the strength of the vortices is even lower due to increased dissipation at the flap surface. Hence, although the blade tip vortices are closer to the flap's surface, which increases their relative impact on the flow close to the wall, the momentum decrease below the second vortex ($x/c \approx 0.86$) is lower than in the baseline configuration due to weaker blade tip vortices and less divergence of the slipstream from the wall. Further downstream, approximately in the middle of the flap at $x/c = 0.9$, the momentum in the boundary-layer is actually increased compared to both the isolated wing and the baseline configuration, as shown in Figure 8.4. This is attributed to the blowing of high-momentum flow along the flap by the propeller, and the relatively weak influence of the blade tip vortices.

The pressures induced by the propeller can be observed on the wing's suction side as a change in pressure coefficient. Comparing the pressure deltas in Figure 8.5 to the ones for the baseline configuration (Fig. 7.26), again shows that the inclined propeller induces less suction on the wing than the horizontally-aligned propeller. The magnitude of the pressure increase behind the propeller is comparable in both configura-

---

**Figure 8.2:** Phase averaged axial velocity and streamlines in the mid-plane at $y/c = 0$ (a). Phase averaged induced pressure and velocities in the mid-plane at $y/c = 0$ (b).
8.1. Aerodynamic interaction phenomena

Figure 8.3: Out-of-plane vorticity $C_{\omega,x}$ and propeller-induced velocities in the $y$-plane at $y/c = 0$ for $\phi = 2^\circ$.

Figure 8.4: Time-averaged boundary-layer over the flap at $x/c = 0.9$, below the propeller axis. The velocity $u$ is parallel to the propeller axis and extracted along the survey-line in Figure 8.3.

The pressure distribution of the wing profile below the propeller axis is shown in Figure 8.6. The main difference to the baseline pressure distribution is that the additional lift ahead of the flap is smaller. Moreover, the pressures on the suction side of the flap, and especially at the trailing edge, are larger in the inclined case. This shows that despite the delayed flow separation over the flap compared to the baseline case, the lift still decreases due to the increased static pressure behind the propeller. Only the suction peak over the curvature of the flap is slightly larger when the propeller is inclined which is due to the propeller being positioned behind the curved part, hence accelerating the flow over the convex area. However, it is futile to consider this a Coanda effect [82] as the suction compared to the simulation without propeller is increased only marginally. This raises the question whether it is possible to exploit a Coanda effect by a propeller behind a curved wall in general, because a possible lift benefit seems to be diminished immediately by the following adverse pressure gradient due to the propeller.

The unsteady behaviour of the pressure distribution in Figure 8.7 is similar to the baseline configuration. The largest suction peak can found when a blade is closest to the wing, at multiples of $\phi = 60^\circ$. Over the entire
chordwise dimension of the flap, however, the suction is largest for phase angles close to $\phi = 10^\circ$, which can also be inferred from the largest pressure drag at that phase angle in Figure 8.8. At this phase angle, the suction over the flap is strong enough to increase the pressure drag compared to the propeller-off configuration. At a phase angle of $\phi = 50^\circ$, the backward facing component of the lift over the flap is so low that the drag of the wing profile is actually negative. Hence, the suction pointing forward at the leading edge is larger than the backward facing component of the lift over the flap. Similar to the baseline configuration, the friction drag of the wing is not affected by the propeller.
8.1. Aerodynamic interaction phenomena

Figure 8.8: Sectional drag coefficient of the wing profile at $y/b = 0$ at different time instances, i.e. for different blade positions.

8.1.2. Flow separation on the wing

While in the baseline configuration flow separation on the flap was just postponed below the outboard part of the slipstream, areas of completely attached flow exist in the inclined configuration. These areas are encircled in Figure 8.9, and are visible as shear-lines going directly from in front of the propeller to the trailing edge of the wing. This is partially attributed to the influence of the overset interface but is predominantly due to the blowing of high-momentum flow by the propeller. At the outer edges of the attached flow regions, two big recirculation cells form due to the locally high momentum flow which creates a velocity gradient to the slower outer regions. The change in pressure coefficient from Figure 8.5, however, showed no significant influence of the cells. Also a view of the pressure distribution on the entire wing’s upper side in the Appendix (Fig. F.1) shows no significant influence of the cells. Behind the propeller, below its axis, the high-momentum flow blown over the flap by the propeller (identified previously in Figure 8.4) follows the flap deflection and curves the outboard streamlines towards the center of the wing. At the propeller location, however, the adverse pressure gradient is large enough to trigger flow separation. Contrary to the reversed flow over the entire flap chord in the middle of the baseline configuration, this separation is followed by a re-attachment of the flow forming a small separation bubble at $(y/c = 0, x/c = 0.825)$.

Figure 8.9: Averaged skin friction coefficient on the wing’s suction side and wall shear-lines. The propeller blades are shown as an example for a phase angle of $\phi = 30^\circ$. The colormap does not span the full range of friction coefficient values.
The time-dependent skin friction coefficient of the middle-wing profile in Figure 8.10(a) exhibits two distinct jumps in skin friction at phase angles of $\phi = 0^\circ$ and $\phi = 10^\circ$. These jumps are due to local flow reversal which was confirmed by checking for a change in sign of the wall-parallel wall shear components. Compared to the averaged line, however, the axial separation point does not move significantly. The averaged line exhibits two points of low skin friction, at about $x/c = 0.82$ and $x/c = 0.83$, which mark the chordwise start and end of the separation bubble. Figure 8.10(b) shows that the beginning of the separation bubble coincides well with the separation point of the baseline configuration. Compared to the propeller-off case, the flow separates more upstream because of the adverse pressure gradient of the propeller. Immediately afterwards, however, the slipstream blown over the flap increases the momentum in the boundary-layer, re-attaching the flow. This is not possible in the baseline configuration due to the larger divergence of the slipstream from the flap. Hence, inclining the propeller with the flap proved to be a valid technique to prevent flow separation by increasing the momentum in the flow close to the flap's surface.

![Graphs showing skin friction coefficients](image)

Figure 8.10: Unsteady and averaged skin friction coefficient on the suction side of the wing profile at $y/c = 0$. In Figure (b), the averaged skin friction coefficient is compared to the skin friction of the wing with nacelle (prop-off), and the baseline configuration.

### 8.2. Propeller loading distributions

The reason why less velocity is induced close to the wing when the propeller is inclined can be found in the thrust distribution over the propeller disc in Figure 8.11. Close to the wing, less thrust is produced, hence less velocity is induced. The non-uniform inflow due to the wing, causes an increase in thrust for the down-going blades between $\phi = 180^\circ$ and $\phi = 300^\circ$, due to an increase in blade angle of attack as predicted from the flow field around the isolated wing (Fig. 6.3(b)). The local thrust increase, however, is not large enough to compensate the relatively weaker decrease in thrust, but over a larger area, such that the total thrust of $C_T = 0.335$ is comparable to the isolated propeller thrust ($C_T = 0.336$). Compared to the baseline thrust ($C_T = 0.341$), the inclined propeller thrust decreased by $\Delta C_T = 0.006$. Figure 8.12 confirms that the inclined propeller produces less thrust close to the wing, at $\phi = 360^\circ$, than the baseline propeller. In the inclined configuration, the prediction of the thrust distribution by the non-linear inflow of the isolated wing is valid, contrary to the baseline case, because the inclined propeller alters the pressure distribution of the wing less than the baseline propeller does.

Although the thrust $T$ defined along the propeller axis exhibits only a small change when the propeller is inclined, the force in flight direction $F_x$ changes more considerably, as the thrust axis is rotated. The propeller-fixed coordinate system with the thrust along the propeller axis and a normal force $N$ perpendicular to it is sketched in Figure 8.13. The resultant force $R$ has a reduced component in x-direction because the backward component of the normal force needs to be compensated. For the baseline propeller, the force in the forward direction $F_x$ is the same as the thrust $T$, and the vertical component $F_z$ of the resultant force is the same as $N$. It becomes obvious that, when the propeller is inclined, not only the horizontal force $F_x$
8.2. Propeller loading distributions

Figure 8.11: Thrust distribution on the propeller disc (a) and difference with respect to the uninstalled case (b).

Figure 8.12: Thrust distribution for different phase angles compared to the isolated blade. The dashed lines represent the thrust distributions of the baseline propeller.
Inclined over-the-wing configuration

8. Inclined over-the-wing configuration

8.1. Aerodynamics

Figure 8.13: Sketch of the forces acting on the inclined propeller. The vectors are not to scale.

Table 8.1: Table comparing the forces acting on the baseline and inclined propeller. Data was taken from zero and 30 degrees phase angle and the mean was calculated to average out the thrust fluctuation which have a wavelength of 60 degrees phase angle.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Baseline</th>
<th>Inclined</th>
<th>Isolated</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\phi$ [deg]</td>
<td>0 30 mean</td>
<td>0 30 mean</td>
<td>-</td>
</tr>
<tr>
<td>$\hat{T}_{c}$</td>
<td>0.0944 0.0944</td>
<td>0.0922 0.0938</td>
<td>0.0930 0.0928</td>
</tr>
<tr>
<td>$\hat{T}_{c,N}$</td>
<td>-0.009 -0.0091</td>
<td>-0.0091 -0.0091</td>
<td>-0.0214 -0.0234</td>
</tr>
<tr>
<td>$\hat{T}_{c,x}$</td>
<td>-0.0944 -0.0944</td>
<td>-0.0944 -0.0944</td>
<td>-0.0794 -0.0801</td>
</tr>
<tr>
<td>$\hat{T}_{c,z}$</td>
<td>-0.009 -0.0091</td>
<td>-0.009 -0.0091</td>
<td>0.0517 0.054</td>
</tr>
<tr>
<td>$\hat{T}_{c,R}$</td>
<td>0.0949 0.0948</td>
<td>0.0948 0.0948</td>
<td>0.0947 0.0966</td>
</tr>
</tbody>
</table>

The forces are represented in this manner here, because they will be used for a balance of forces with a wing segment of 1.1 times the propeller diameter $D$, in a later section. The subscripts $N, x, z$ and $R$ indicate which force component is taken. The results in Table 8.1 confirm that the thrust in the theoretical flight direction, $\hat{T}_{c,x}$, decreases when the propeller is inclined, while the vertical component, $\hat{T}_{c,z}$, increases due to the thrust vectoring. Moreover, the vertical component increases more than the horizontal one decreases, which causes an increase in resultant thrust, $\hat{T}_{c,R}$, when the propeller is inclined.

Similar to the thrust, also the torque and shaft power of the inclined propeller ($C_Q = 0.1029$, $C_P = 0.647$) change only slightly compared to both the baseline results ($C_Q = 0.1022$, $C_P = 0.642$) and the isolated values ($C_Q = 0.1023$, $C_P = 0.643$). The difference in torque coefficient between the inclined and baseline configuration is less than the uncertainty in the torque coefficient of $U_{CQ} = \pm 0.0017$, established in Section 4.1.2. In the following sections, it will be shown how this can be exploited to achieve different wing loads by thrust vectoring.

8.3. Wing loads

Due to the relatively low increase in suction on the wing, despite the delay of flow separation, the lift distribution in Figure 8.14 shows only a moderate increase in the middle. Compared to the peak in spanwise lift of the baseline configuration ($C_l = 0.15$), the peak in the inclined configuration is approximately 24% lower ($\Delta C_l = -0.03$). The drag, however, is decreased more than in the baseline configuration, due to the decreased suction over the flap and postponed flow separation, which results in a similar lift-to-drag ratio. The post-
8.3. Wing loads

Figure 8.14: Spanwise lift, drag, and lift-to-drag ratio of the wing, averaged over 60 degrees of propeller rotation

Figure 8.15: Spanwise lift and drag of the wing for different propeller phase angles.

poned flow separation does not significantly benefit the lift since the inclined propeller also increases the static pressure directly above the flap, which decreases the lift. For the drag, however, both the postponed flow separation and the increased static pressures over the flap are beneficial as they decrease the pressure drag. Hence, the inclined configuration is more beneficial for reducing drag on the wing than the baseline one, while in the baseline configuration the lift increase on the wing is larger due to the stronger propeller-induced suction on the wing. Since the inclined propeller does not have the same asymmetry in thrust (Fig. 8.11(a)) between the up-going and down-going blades close to the wing, as the baseline propeller does (Fig. 7.35(a)), there is no significant asymmetry in the spanwise drag distribution, contrary to the baseline case.

The unsteady lift distributions in Figure 8.15 show that the influence of the propeller extends beyond its diameter because of the areas of postponed flow separation on the wing. In these areas the relative contribution of the postponed flow separation to the lift enhancement is increased, since the negative contribution of the increased static pressure behind the propeller is less. The highest peaks in the lift distributions, and with this also the highest peaks in drag due to the increased backward-facing suction, follow the movement of the blades above the flap.

Similar to the baseline configuration, the difference between propeller-on, and propeller-off, lift and drag can be integrated for various wing segments to obtain the induced lift and drag deltas. This is shown in Figure 8.16 for both the inclined and baseline configuration. Both the lift enhancement and drag reduction decrease with increasing span of the wing segment, and it can be confirmed that for all wingspans shown, the drag, but also the lift, decrease in the inclined case with respect to the baseline configuration. Contrary to the baseline wing, the lowest drag in the inclined case is obtained for the smallest wing segments, due to
the more symmetric drag distribution. It should be noted that the lift and drag mentioned here are the ones of the wing influenced by the propeller. Hence, the forces on the propeller need to be taken into account as well, to derive realistic conclusions regarding the lift and drag of the combined system. This will be done in the next section for a wing segment of 1.1\(D_p\) span.

### 8.4. Comparison between baseline and inclined system performance

Because of the thrust vectoring of the inclined propeller, the lift of the combined system has a larger vertical component, \(\hat{T}_{c,z}\), of the resultant force, than the baseline configuration. Table 8.2 shows that this additional force compensates for the weaker lift increase of the wing segment (\(\Delta C_L\)), such that the total force in the vertical direction is 24% larger than for the baseline propeller configuration. The inclination of the propeller, however, also leads to a weaker additional forward force, \(\hat{T}_{c,x}\), to the system. This reduction in net thrust is not compensated by the decrease in drag of the wing (\(\Delta C_D\)), as can be inferred from Table 8.3. Hence, the baseline configuration has a 20% larger installed thrust than the inclined configuration.

Table 8.2: Balance of forces in z-direction for a wing-segment span of \(b'/D = 1.1\) and relative change with respect to the baseline case.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>(C_{iso}^{\text{np}}) [-]</th>
<th>(\Delta C_L) [-]</th>
<th>(\hat{T}_{c,z}) [-]</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>0.093</td>
<td>0.050</td>
<td>-0.009</td>
<td>0.134</td>
</tr>
<tr>
<td>Inclined</td>
<td>0.093</td>
<td>0.020</td>
<td>0.053</td>
<td>0.166</td>
</tr>
</tbody>
</table>

Table 8.3: Balance of forces in x-direction for a wing-segment span of \(b'/D = 1.1\) and relative change with respect to the baseline case. The thrust is positive in the negative x-direction.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>(C_{iso}^{\text{np}}) [-]</th>
<th>(\Delta C_D) [-]</th>
<th>(\hat{T}_{c,x}) [-]</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>0.0137</td>
<td>-0.0018</td>
<td>-0.0944</td>
<td>-0.0825</td>
</tr>
<tr>
<td>Inclined</td>
<td>0.0137</td>
<td>-0.0025</td>
<td>-0.0797</td>
<td>-0.0685</td>
</tr>
</tbody>
</table>

Although the shaft power differs slightly between the configurations, it is assumed to be constant due to a relatively low standard deviation of the power coefficient of \(\sigma = 0.0026\) between the values. This assumption is used to show how the propulsive efficiency of an OTW-propeller-system can increase compared to an uninstalled configuration, if the installed thrust increases. To evaluate the efficiency of such an integrated system, the propulsive efficiency is defined with the installed thrust as:

\[
\eta_{\text{pro}} = \frac{T_{x,\text{inst}} \cdot V}{P_{s}} \tag{8.2}
\]
where $T_{\text{inst}} = |F_x| - \Delta D$, with $F_x$ being the force generated by the propeller in the $x$-direction. Since the thrust, and with this $|F_x|$, of the baseline OTW propeller is slightly higher than the thrust of the isolated propeller at the same shaft power, the drag saved on the wing leads to a higher installed thrust. Hence, the propulsive efficiency increases when the propeller is installed. Also the corrected lift to drag ratio (Equation 7.13) of the baseline configuration increases, due to the inclusion of the positive change in propeller thrust, compared to an unintegrated design at the same lift coefficient. This contradicts the claim by Müller et al. [47] that "the best aerodynamic performance is achieved for configurations without interaction". However, crucial here is that the propeller thrust stayed practically constant, whereas typically a decrease in thrust of up to 20% is observed [40, 47].

Table 8.4 compares the resulting propulsive efficiencies for a wing-segment of $1.1D_p$ span. The baseline configuration has the largest propulsive efficiency due to its largest installed thrust, enabled by the non-negative change in propeller thrust and reduction in drag of the wing. Despite creating the largest reduction in drag of the wing, the inclined configuration has the lowest installed thrust due to the inclination of the propeller-axis, hence, the lowest propulsive efficiency. The smallest change in propulsive efficiency ($0.61/0.59 = 1.034$) is an order of magnitude larger than the smallest change in power coefficient ($0.643/0.642 = 1.002$). Hence, the assumption of a constant power coefficient only has a small effect on the change in propulsive efficiency.

Table 8.4: Table comparing the propulsive efficiency.

| Configuration | $C_P$ [-] | $|T_{c,x}|$ [-] | $T_{\text{inst}}$ [-] | $\eta_{\text{pro}}$ [-] |
|---------------|-----------|-----------------|-----------------|-----------------|
| Isolated      | 0.643     | 0.0928          | 0.0928          | 0.59            |
| Baseline      | 0.642     | 0.0944          | 0.0961          | 0.61            |
| Inclined      | 0.647     | 0.0797          | 0.0821          | 0.52            |
Conclusions and recommendations
Conclusions

In this thesis, the aerodynamic interaction between a wing and an over-the-wing propeller was studied by analysing the results of unsteady RANS simulations. First, a baseline configuration was evaluated by comparing the numerical results to experimental ones, identifying the influence of the wind tunnel walls on the aerodynamic interaction, studying the characteristic aerodynamic interaction effects, and explaining their influence on the propeller loading distributions and the wing loads. Subsequently, an inclined propeller configuration intended to postpone flow separation on the flap was investigated, starting with the aerodynamic interaction phenomena, followed by the resulting propeller loading distributions, wing loads, and a comparison between the two systems regarding their propulsive efficiency. The methodology of adding the respective force coefficients of a wing segment and OTW propeller, demonstrated for the wing in this thesis, can easily be applied to a more realistic wing profile for high lift in the development process of a distributed OTW propeller system.

The aerodynamics of the isolated wing and the thrust of the installed propeller were captured reasonably well, compared to the experimental results. Furthermore, the blade tip vortices were visible as well as the pressures induced by the propeller together with the resulting propeller-induced flow separation on the flap. However, the uncertainty in the numerical results is relatively high since the magnitude of the propeller-induced flow separation was not captured accurately. This was because of the influence of the overset interface on flow gradients in the boundary-layer, the modelling of turbulence used by the RANS solver, and most importantly, an underestimation of the blade tip vortex intensity. The blade tip vortices were visible in the numerical results, however, with a decreased strength in vorticity due to numerical diffusion. Therefore, RANS simulations can be used to study trends for an OTW propeller configuration provided that the calculations are performed unsteadily to capture the blade tip vortices and the mesh is sufficiently refined to mitigate numerical diffusion.

The inaccurate prediction of flow separation led to a 45 % higher sectional pressure-lift coefficient, and 43 % lower pressure-drag coefficient, below the propeller axis, compared to the experimental results. It should be noted, however, that these differences are relatively large due to the low absolute values of the lift and drag coefficients of the wing. The absolute differences in lift and drag coefficient compared to the experiment were $\Delta C_{l,p} = 0.052$ and $\Delta C_{d,p} = 0.0038$, respectively. Moreover, the results revealed that the wind tunnel walls increase the total lift and drag of the wing due to blockage and interference with the boundary-layer of the wing. However, for the wing profile below the propeller axis, a decrease in pressure drag due to the walls could be identified, which was caused by slipstream blockage. This resulted in a larger propeller-induced drag reduction with walls than without. The propeller-induced lift increase of the wing profile showed no significant difference between the case with wind tunnel walls, and without. Similarly, the flow separation over the flap was practically unaffected.

The results of the baseline configuration showed that the propeller generated a low-pressure region on the wing surface in front of it. At the leading edge of the wing, only a weak suction increase was identified as it was relatively far away from the propeller location. Directly behind the propeller, the pressure on the wing surface was increased due to the static-pressure jump at the propeller location and the interaction of the slipstream and the blade tip vortices with the flap. The resulting adverse pressure gradient did not immediately cause flow separation at the propeller location but only weakened the boundary-layer, such that flow separated on the flap further downstream, where the slipstream diverging from the deflected flap and the blade tip
vortices caused a momentum deficit in the boundary-layer of the flap. However, this adverse effect of the propeller slipstream on the momentum in the boundary-layer was dominant only below the propeller axis, where the slipstream, and also the blade tip vortices, were closest to the wing. At this spanwise location, a temporal dependency of the point of flow separation on the unsteady shedding of the blade tip vortices could be identified. In front of the propeller, a favourable pressure gradient was present which increased the momentum in the boundary-layer to overcome the adverse pressure gradient of the flap below the outboard area of the propeller blades and postpone flow separation where the slipstream was not closest to the wing.

The inclined propeller induced less suction on the wing because of decreased thrust close to the wing caused by the wing's non-uniform inflow. Nevertheless, the inclined propeller was able to locally re-attach separated flow on the flap, below the propeller axis, by blowing a high-momentum flow over the flap. The point of flow separation below the propeller axis, with respect to the wing, was independent of the propeller inclination. Due to the fowler motion of the inclined propeller, however, the inclined propeller was directly above the point of flow separation whereas the baseline propeller was in front of it. Comparing the slipstream-flap interaction between the baseline and inclined configuration showed that the slipstream can either cause flow separation, if it diverges from the flap, or re-attach separated flow if it stays close to the flap's surface. However, the postponed flow separation in the inclined case was not able to increase the lift over the flap compared to the baseline case because of the increased static pressure in the propeller's slipstream. It did, however, decrease the pressure drag of the wing.

The total propeller thrust was not significantly affected by the presence of the wing for both configurations. Only the azimuthal distributions of thrust over the propeller disc changed due to the non-uniform inflow that was caused by the wing-induced velocities, plus the inclination of the propeller in the inclined case. The non-uniformity of the propeller inflow can be split into two contributions: the speed of the inflow affecting the axial blade velocities, and the direction of the inflow velocity affecting the tangential blade velocities. In the baseline configuration, the change to the direction of the inflow velocity had a larger impact on the thrust distribution than the local increase in flow speed close to the wing, whereas in the inclined configuration both contributions were practically equal in intensity. This resulted in a thrust coefficient of the baseline configuration that was comparable to the isolated propeller thrust and slightly higher than the inclined propeller thrust.

The lift and drag of the wing were very sensitive to changes in the pressure over the flap. Even a small increase in suction over the flap was enough to increase the pressure drag noticeably. Hence, drag was reduced rather by increasing the pressure on the flap than increasing suction at the leading edge because of its relative distance to the propeller. The lift of the wing increased the most in the baseline configuration due to the stronger induced suction compared to the inclined propeller. However, if only a small wing-segment with the span in the order of the propeller diameter is considered as a combined system with the propeller, then the additional vertical force component of the inclined propeller could compensate the lower lift of the wing. The drag on the wing was reduced the most with the inclined propeller due to the increased pressure in the slipstream and postponed flow separation. However, for the same small wing-segment-system, the baseline configuration resulted in the largest installed thrust as it had a larger forward contribution of the propeller forces than the inclined configuration.

The results in this thesis showed that an over-the-wing propeller can increase the propulsive efficiency of the system, at a constant shaft power, compared to a decoupled propulsor-wing system and, therefore, increase its fuel efficiency. However, this was only possible in the baseline configuration due to the larger forward component of the thrust vector than in the inclined configuration and the practically unaffected magnitude of the propeller thrust. Hence, during the development process of such an OTW system, the wing and propeller must be designed in a synergy such that penalties on the propeller thrust can be prevented when mounted above the wing. Moreover, blade tip vortices and their influence on the boundary-layer of the wing need to be captured sufficiently to accurately predict flow separation.
This study has provided significant insight into the aerodynamic interaction effects between an OTW propeller and a deflected flap, the capability of an inclined OTW propeller to prevent flow separation on the flap, and the relevance of modelling the tip vortices accurately to correctly predict propeller-induced flow separation. However, several aspects of the numerical setup can be improved, and directions for future research can be established.

The use of an overset mesh reduced the effort to create the second, inclined, configuration as the propeller mesh just had to be inserted at a different location and angle to the wing, while the surrounding wing mesh stayed the same. The generation of the propeller mesh, however, was not as straightforward as expected. While discretising the propeller domain, one had to constantly keep in mind where the propeller will be placed relative to the wing, to ensure a sufficient mesh overlap. This became particularly challenging in the small gap between the propeller tips and along the flap. The resulting step to a structured mesh for improved overlap likely further aggravated the numerical diffusion process. Also the overset interface itself turned out to cause an inaccurate prediction of flow separation due to the interpolation of flow gradient. Thus, it should be avoided to place an overset interface in a region where high flow gradients are expected like in a boundary-layer or within the area of influence of vortices. For an OTW propeller application, however, this is nearly impossible. Hence, a control simulation featuring an empty overset mesh should ideally be performed for every configuration that includes an overset interface to verify the results. Nevertheless, once a working propeller mesh is established, it can easily be placed at different locations relative to a wing, with only minor mesh refinements necessary. This is a clear advantage over a sliding mesh technique, for instance, for design studies where many different configurations need to be evaluated since the background mesh requires no adaption. Furthermore, the comparison to the wind tunnel measurements highlighted the importance of having experimental data for validation to assess the uncertainty in the numerical results. For future numerical studies of an OTW configuration, it is therefore highly recommended to obtain experimental data as well, at least of a basic control configuration to compare the propeller induced flow separation and blade tip vorticity.

The calculations of the propeller-induced lift and drag for different wing segments were made under the assumption that adjacent propellers do not change the trend in lift increase, and drag decrease. Therefore, it needs to be studied whether having multiple propellers in close proximity only augment the benefits of OTW propulsion or possibly introduce detrimental effects such as increased flow separation on the flap or decreased propeller thrust due to the interaction of the adjacent slipstreams. However, the lift and drag of the wing strongly depend on the airfoil of the wing, thus, a similar experimental or unsteady RANS study with a more realistic airfoil is necessary as well to confirm the trends in lift, drag, and propulsive efficiency of a single OTW propeller for a practical aircraft application. Thereby, a focus on the propeller thrust is key as the increased propulsive efficiency of the baseline configuration was only possible due to the small increase in the OTW propeller thrust. In this context, an optimisation study of the wing profile could lead to an improved lift and drag of the wing while minimising a performance penalty on the propeller. The present study already suggests that this could be done by increasing the tangential blade velocities while keeping the increase in axial velocity minimum. Similarly, a sensitivity study of the propeller inclination could be used to find an optimum between induced suction on the wing by the propeller and postponed flow separation on the flap by the slipstream. Particularly interesting would be to find out if an inclined propeller at the same chord-
wise position of the baseline propeller can still postpone flow separation, and at what inclination angle. In that context, a slightly inclined propeller at the baseline propeller location might exhibit a further increase in thrust compared to the baseline propeller due to the angle of the inflow, thereby further increase the magnitude of the induced pressures, while possibly postponing flow separation on the flap better than the baseline configuration, but less than the inclined configuration in this thesis.

Out of the scope of this thesis, but still relevant for the development process of an OTW propeller system, are also studies regarding the structural feasibility of such a design. Particularly for the inclined configuration, where the propeller is attached to the flap, structural challenges arise due to additional moments. It needs to be ensured that potential lift gains are not compromised for by an increase in weight of the wing structure or power train. Furthermore, the non-uniform thrust distribution in both the baseline and inclined case will likely cause vibrations and additional noise. For a wing featuring an inclined OTW propeller, the noise emitted is targeted more towards the ground and, hence, the noise shielding capability would need to be re-assessed.
Additional results
Isolated propeller performance

The same propeller as in this thesis, is also used in Ref. [70], where an experimental and numerical study of its isolated performance is presented. The propeller blades feature a Clark-Y airfoil family with distributions in chord length and blade pitch angle as shown in Figure A.1. The performance curve in Figure A.2 indicates for a free-stream velocity of $V_\infty = 20$ m/s a thrust coefficient of $C_T = 0.35$ at an advance ratio of $J = 1.1$. This confirms that also in the experiments for this thesis, not only in the numerical results, the thrust of the installed propeller (Fig. 7.11) deviates negligibly from the isolated thrust value. A decrease in thrust compared to the isolated propeller can only be identified at higher advance ratios.

Figure A.1: Chord distribution and blade pitch angle of a propeller blade, taken from Ref. [70].

Figure A.2: Isolated propeller performance curves obtained from numerical and experimental data in Ref. [70].
Overset influence on velocity gradients

Particular attention regarding an appropriate match in cell size was paid for the cells in the propeller tip region shown in Figure B.1. The outer structured cells of the propeller mesh have approximately the same dimensions as the outer most cells of the wing mesh, such that no sharp jumps in the velocity gradient can be observed. Nevertheless, interpolating the relatively large flow gradients inside the boundary-layer across the overset interface may result in an inaccurate prediction the boundary-layer height, and the influence of the blade tip vortices on the boundary-layer gets reduced. However, for the isolated wing it was shown in Figure 4.13 that the absolute height of the boundary-layer is affected only marginally.

Over the flap, on the other hand, the cells in the inflation layer of the wing do not match well with the structured cells of the propeller mesh. This causes a region of increased velocity gradient along the overset interface as shown in Figure B.2. The colourmap in this figure features negative values, contrary to the previous one, such that the influence of the overset interface on the velocity gradient on the upper side of the flap can be compared to the velocity gradient on the bottom side of the flap, where no interface is present. Similarly, cells of the cylindrical propeller mesh that penetrate the wing boundary-layer cause local irregularities in the flow gradients as shown in the rear view of the propeller plane in Figure B.3.

Figure B.1: Vertical gradient of the axial velocity in the plane along the propeller axis, zoomed into the tip region.
Figure B.2: Vertical gradient of the axial velocity in the plane along the propeller axis.

Figure B.3: Vertical gradient of the axial velocity in the propeller plane, viewed from behind the propeller.
Vortex shedding behind wing in the wind tunnel

Figure C.1 (a) shows the out-of-plane vorticity in a plane at $y/c = 0$ in the propeller-off case, hence, without propeller blades. The Q Criterion shows no vortex cores behind the wing, except for a small one at the trailing edge of the wing due to flow separation on the flap. When the propeller is on (Fig. C.1 (b)), a systematic arrangement of vortex cores are predicted by the Q Criterion behind the wing.

The cores identified by the Q Criterion are approximately $x/c = 0.2$, equal to 0.21 m, apart. At a free-stream velocity of $V_\infty = 20$ m/s, each structure is 0.011 seconds apart. This is equal to 0.96 propeller revolutions at the propeller rps of 87 Hz. Hence, it is possible that there is a connection between the fluctuations in lift and these vortical structures. However, this calculation is just a rough estimation of the time between the vortex, and not to be taken as exact, since the velocity in the wake is not constant and less than the free-stream velocity, and the distance between the centers appears to increase. Furthermore, it is difficult to asses whether this is something that happens in reality or is just a numerical instability of the solver. Moreover, it is unclear whether the structures identified by the Q Criterion are a cause or effect of the fluctuations in lift. However, wall-effects on the shedding frequency behind a cylinder with a similar blockage ratio, but smaller diameter-based Reynolds number compared to a thickness-based Reynolds number of the wing, are identified in a similar range of frequencies in Ref [6].
Figure C.1: Vorticity in [1/s] and Q-Criterion in a plane at $y/c = 0$ for the baseline propeller-off and propeller-on case inside the wind tunnel. The colourbars do not span the full range of vorticity to improve readability.
Baseline thrust distribution with wind tunnel walls

The purpose of Figure D.1 is to confirm that not only the total thrust of the propeller is unaffected by the wind tunnel walls but also the thrust distribution.

Figure D.1: Thrust distribution in the baseline configuration with wind tunnel walls.
Change in skin friction on the baseline wing

Figure E.1 shows the difference in skin friction between the propeller-on case and the propeller-off case. In front of the propeller, skin friction increases due to the flow acceleration by the propeller. At the propeller location, skin friction decreases as the slipstream induces a reduction in velocity over the flap curvature, until it increases again over the flap due to postponed flow separation.

Figure E.1: Difference in skin friction on the wing between propeller on and propeller off.
Streamlines on the wing with inclined propeller

Figure F.1 shows that the circulation cells peripheral to the inclined propeller have no significant influence on the wing pressure distribution.

Figure E1: Pressure coefficient and streamlines on the suction side of the wing.


