An Algorithm for the Aerodynamic Analysis of Rotating Blades Using a Weak Viscous-Inviscid Interaction Method

by

Mengu Umut Tinis

A thesis submitted to the Faculty of Graduate and Postdoctoral
Affairs in partial fulfillment of the requirements for the degree of

Master of Applied Science

in

Aerospace Engineering

Carleton University
Ottawa, Ontario

© 2013

Mengu Umut Tinis

Library and Archives
Canada

Published Heritage
Branch

395 Wellington Street
Ottawa ON K1A 0N4
Canada

Bibliothèque et
Archives Canada

Direction du
Patrimoine de l'édition

395, rue Wellington
Ottawa ON K1A 0N4
Canada

Your file Votre référence

ISBN: 978-0-494-94654-1

Our file Notre référence

ISBN: 978-0-494-94654-1

NOTICE:

The author has granted a non-exclusive license allowing Library and Archives Canada to reproduce, publish, archive, preserve, conserve, communicate to the public by telecommunication or on the Internet, loan, distribute and sell theses worldwide, for commercial or non-commercial purposes, in microform, paper, electronic and/or any other formats.

The author retains copyright ownership and moral rights in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author's permission.

AVIS:

L'auteur a accordé une licence non exclusive permettant à la Bibliothèque et Archives Canada de reproduire, publier, archiver, sauvegarder, conserver, transmettre au public par télécommunication ou par l'Internet, prêter, distribuer et vendre des thèses partout dans le monde, à des fins commerciales ou autres, sur support microforme, papier, électronique et/ou autres formats.

L'auteur conserve la propriété du droit d'auteur et des droits moraux qui protègent cette thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

In compliance with the Canadian Privacy Act some supporting forms may have been removed from this thesis.

While these forms may be included in the document page count, their removal does not represent any loss of content from the thesis.

Conformément à la loi canadienne sur la protection de la vie privée, quelques formulaires secondaires ont été enlevés de cette thèse.

Bien que ces formulaires aient inclus dans la pagination, il n'y aura aucun contenu manquant.

Canada

Abstract

An algorithm based on a weak viscous-inviscid interaction method is developed for the aerodynamic analysis of rotating blades under steady and incompressible flow conditions for configurations where the aerodynamics of each blade can be treated in isolation. A weak viscous-inviscid interaction method for calculating the aerodynamics of airfoil geometries, which is based on a two-dimensional source-vortex panel method and the momentum-integral method, is updated using state-of-the-art models and correlations to predict specific flow phenomena such as transitioning flow and separated flow. The blade-element-momentum theory with Prandtl root- and tip-loss corrections is used to calculate the three-dimensional aerodynamics of rotating blades, using the two-dimensional aerodynamic predictions of the viscous-inviscid interaction method and empirical stall delay models. The algorithm is shown to provide predictions that are sufficiently accurate for blade design optimization under unstalled operating conditions. The accuracy of the algorithm is limited to about $3^\circ \sim 5^\circ$ angle of attack beyond stall.

Acknowledgements

First and foremost, I would like to thank my thesis supervisors, Drs. Metin I. Yaras, Fred F. Afagh, and Robert G. Langlois for providing me with this wonderful opportunity to work in my area of interest and on a project which offers unlimited learning opportunities. I thank Dr. Yaras for his constant guidance, motivation, and advice, which have been invaluable throughout this study. The many hours of discussion we had are greatly appreciated. I would like to thank Drs. Afagh and Langlois for staying on board and supporting this study until completion even after the focus of the study shifted away from their primary area of interest. I would also like to acknowledge all my colleagues and friends in Dr. Yaras' research group for their support and feedback during my studies, and particularly Joshua Brinkerhoff for his instrumental suggestions to help me bring this manuscript to completion.

My greatest thanks go to my father, Rait Tinis, my mother, Fusun Tinis, and my better half, Esra Merdimeydan. Without their love, support, and encouragement, this accomplishment would not have been possible.

Table of Contents

Abstract.....	ii
Acknowledgements	iii
Table of Contents.....	iv
List of Tables	x
List of Figures	xi
Nomenclature.....	xv
Chapter 1: Introduction	1
1.1 Motivation	1
1.2 Objectives.....	2
1.3 Approach	3
Chapter 2: Background Information on Rotating Blade Aerodynamics	4
2.1 Aerodynamic Working Principle of Wind Turbine and Propeller Blades	4
2.2 Methods for Calculating Turbomachinery Performance	6
2.2.1 Momentum (Actuator Disc) Theory	7
2.2.2 Blade-Element Theory (BET).....	12
2.2.3 Blade-Element-Momentum Theory (BEMT)	15
2.3 Governing Equations of Fluid Dynamics.....	18
2.4 Inviscid Flow	19
2.4.1 Euler Equations	19
2.4.2 Potential Flow Equations	20
2.4.3 Potential-Flow-Singularity Methods	21
2.5 Viscous Flow	22

2.5.1 Boundary-Layer Equations	22
2.5.2 The Momentum-Integral Relation.....	26
2.5.2.1 Laminar Closure Method.....	28
2.5.2.2 Turbulent Closure Method.....	29
Chapter 3: Literature Review	32
3.1 Analysis Methods for Turbomachinery Performance and Aerodynamics.....	32
3.1.1 Momentum (Actuator Disc) Theory.....	32
3.1.2 Blade-Element Theory (BET).....	33
3.1.3 Blade-Element-Momentum Theory (BEMT)	34
3.1.4 Navier-Stokes Equation Solutions	36
3.1.5 Viscous-Inviscid Interaction (VII)	38
3.2 Inviscid Flow Analysis Methods	45
3.2.1 Thin Airfoil Theory (TAT).....	46
3.2.2 Lifting-Line Blade Model.....	47
3.2.3 Surface Singularity (Panel) Methods	49
3.2.3.1 3D Panel Methods.....	52
3.2.3.2 2D Panel Methods.....	54
3.3 Viscous Flow Analysis Methods	54
3.3.1 Similarity Solutions for the Boundary-Layer Equations	55
3.3.2 Alternative Analysis Methods for Boundary Layers.....	57
3.3.2.1 Integral Methods	58
3.3.3 Flow Transition	60
3.3.3.1 Modes of Flow Transition	60

3.3.3.2	Factors Influencing Flow Transition	62
3.3.3.3	Transition Models	63
3.3.4	Flow Separation	66
3.4	Additional Aerodynamic Effects for Consideration	68
3.4.1	Lift-Induced Velocity	68
3.4.2	Lift Loss at the Blade Root and Tip	68
3.4.3	Unsteady Aerodynamic Effects.....	69
3.4.3.1	Corrigan-Schillings Stall Delay Model	71
3.4.3.2	“Centrifugal Pumping” Model of Lindenburg	73
3.4.4	Rotor Wake Effects	74
3.4.5	Wind Shear.....	76
3.4.6	Streamwise Surface Curvature.....	78
Chapter 4:	Algorithm for the Aerodynamic Analysis of Rotating Blades.....	80
4.1	Introduction	80
4.2	A Weak Viscous-Inviscid Interaction Method for Steady, 2D, Incompressible Flow around an Airfoil.....	80
4.2.1	Inviscid Flow Analysis Using a 2D Source-Vortex Panel Method.....	81
4.2.1.1	Discretization for the 2D Panel Method.....	81
4.2.1.2	Boundary Conditions.....	83
4.2.1.3	System of Equations.....	87
4.2.1.4	Inviscid Flow Velocity and Pressure Coefficient Distribution	89
4.2.1.5	Sizing of Airfoil Surface Panels.....	90
4.2.2	Viscous Flow Analysis Using the Momentum-Integral Relation	92

4.2.2.1	The Momentum-Integral Relation.....	92
4.2.2.2	Solution Grid for the Viscous Flow Analysis	94
4.2.2.3	Laminar Momentum-Integral Calculation.....	95
4.2.2.4	Turbulent Momentum-Integral Calculation.....	97
4.2.2.5	Transition Model	100
4.2.2.6	Calculation of the Boundary-Layer Parameters.....	105
4.2.2.6.1	Calculation of the Boundary-Layer Parameters in an Attached- Flow Transition Region	105
4.2.2.6.2	Calculation of the Boundary-Layer Parameters in a Separated-Flow Transition Region	107
4.2.2.7	Treatment of Flow Separation	109
4.2.3	Calculation of the 2D Aerodynamic Coefficients.....	113
4.2.4	Interfacing of the Viscous and Inviscid Flow Solutions.....	116
4.3	Calculation of the 3D Aerodynamics of Rotating Blades	119
4.3.1	Application of the Blade-Element-Momentum Theory for the 3D Aerodynamic Analysis of Wind Turbine Blades	120
4.3.1.1	Calculation of Induced Velocity for Wind Turbine Blades.....	121
4.3.1.2	Calculation of the Effective Angle of Attack	123
4.3.2	Application of the Blade-Element-Momentum Theory for the 3D Aerodynamic Analysis of Propeller Blades	124
4.3.2.1	Calculation of the Induced Velocity in Propeller Blades	124
4.3.2.2	Calculation of the Effective Angle of Attack	127
4.3.3	Stall Delay Corrections.....	128
4.4	Calculation of Rotor Performance	129

Chapter 5: Verification and Validation	131
5.1 Inviscid Flow Analysis Algorithm.....	131
5.2 Viscous flow Analysis Algorithm.....	132
5.2.1 Laminar Momentum-Integral Predictions	132
5.2.2 Turbulent Momentum-Integral Predictions	133
5.2.3 Boundary Layers with Attached-Flow Transition.....	136
5.3 Weak Viscous-Inviscid Interaction Algorithm for Predicting 2D Airfoil Performance.....	139
5.3.1 S809 Airfoil	140
5.3.1.1 Airfoil Surface Pressure Distributions	140
5.3.1.2 Lift Coefficient.....	144
5.3.1.3 Drag Coefficient.....	146
5.3.1.4 Pitching-Moment Coefficient	147
5.3.2 NACA 0012 Airfoil.....	148
5.3.2.1 Airfoil Surface Pressure Distributions	148
5.3.2.2 Lift Coefficient.....	150
5.3.2.3 Drag Coefficient.....	150
5.3.2.4 Pitching-Moment Coefficient	151
5.4 Aerodynamic Loads and Performance of a Rotating Wind Turbine Blade	152
5.4.1 Normal and Tangential Forces.....	154
5.4.2 Pitching Moment.....	158
5.5 Computation Time	159
Chapter 6: Conclusions and Future Work	161

6.1	Conclusions	161
6.2	Future Work.....	162
Appendices		163
Appendix A: Flowchart of the Present Aerodynamic Analysis Algorithm		163
Appendix B: Coordinates for the S809 and NACA 0012 Airfoils		167
B.1	S809 Airfoil	167
B.2	NACA 0012 Airfoil.....	170
References		175

List of Tables

Table 3.1: Atmospheric boundary-layer model coefficients.....	77
Table 4.1: Experimental data used for estimating the displacement thickness within separation bubbles.	108
Table 5.1: Test cases for the verification and validation of the 2D viscous-inviscid interaction algorithm.....	139
Table 5.2: Transition mode and nondimensional chordwise separation, transition inception, and transition completion locations for Case S2 at $\alpha = 5.1^\circ$	141
Table 5.3: Transition mode and nondimensional chordwise separation, transition inception, and transition completion locations for Case S2 at $\alpha = 10.2^\circ$	142
Table 5.4: Transition mode and nondimensional chordwise separation, transition inception, and transition completion locations for Case S2 at $\alpha = 20.2^\circ$	144
Table 5.5: Rotor data and operating conditions for the NREL test turbine.	153
Table 5.6: Blade geometry of the NREL test turbine (lift-generating sections only).	153
Table B.1: Set of S809 airfoil coordinates provided by Somers (1997) with extra coordinates calculated in the present study using a piecewise cubic Hermite interpolating polynomial shaded in gray.	167
Table B.2: Set of NACA 0012 airfoil coordinates provided by Harris (1981) with extra coordinates calculated in the present study using a piecewise cubic Hermite interpolating polynomial shaded in gray.	170

List of Figures

Figure 2.1: Velocities and aerodynamic forces on a wind turbine blade.....	5
Figure 2.2: Momentum theory model of a wind turbine.	7
Figure 2.3: Blade-element model for a wind turbine operating in axial (unyawed) flow.	12
Figure 3.1: Subdivision of the flowfield around an airfoil, with an inviscid outer region and a viscous shear layer (exaggerated in thickness).	39
Figure 3.2: Viscous-inviscid interaction scheme for the direct method.	40
Figure 3.3: Viscous-inviscid interaction schemes for the (a) inverse method and (b) semi- inverse method.	42
Figure 3.4: (a) Generic lifting-line model for a finite wing. (b) Lifting-line blade model by Prandtl (1918).....	48
Figure 4.1: Terminology for a 2D panel method.	82
Figure 4.2: Discretization of the streamwise distance along the upper and lower surfaces for the momentum-integral analysis.	93
Figure 4.3: Schematic of a shear layer (thickness exaggerated) leaving the airfoil surface after flow separation.	111
Figure 4.4: Correlation between stalled angles of attack and tangent coefficient, m , for the (a) S809 and the (b) NACA 0012 airfoils.	112
Figure 4.5: Error of the predicted C_l at higher- and lower-than-optimum values of m for the (a) S809 and the (b) NACA 0012 airfoils.	112

Figure 4.6: Cross-sectional view of a wind turbine blade with aerodynamic force and moment coefficients.	120
Figure 4.7: Propeller blade element.....	124
Figure 5.1: C_p distribution on one half of a 2D cylinder in cross-flow.	132
Figure 5.2: Comparison of the present laminar momentum-integral predictions against (a) Blasius' similarity solution and (b) the similarity solution for a Falkner-Skan self-similar flow past a 27° wedge.....	133
Figure 5.3: Comparison of predicted C_f against Prandtl's power-law approximation for a zero-pressure-gradient turbulent boundary layer.....	134
Figure 5.4: Prediction of C_f in a zero-pressure-gradient turbulent boundary layer.	134
Figure 5.5: Prediction of Re_θ and Re_δ in a zero-pressure-gradient turbulent boundary layer.	135
Figure 5.6: Prediction of Re_θ for boundary layers with attached-flow transition.....	136
Figure 5.7: Prediction of C_f for boundary layers with attached-flow transition.....	137
Figure 5.8: Prediction of H for boundary layers with attached-flow transition.	138
Figure 5.9: C_p distribution for Case S2 at $\alpha = 5.1^\circ$	140
Figure 5.10: C_p distribution for Case S2 at $\alpha = 10.2^\circ$	142
Figure 5.11: C_p distribution for Case S2 at $\alpha = 20.2^\circ$	143
Figure 5.12: C_l versus α for the S809 airfoil for Cases S1 and S2.....	145
Figure 5.13: C_l vs. α for Case S2, using uniform- C_p approach downstream of flow separation.	146

Figure 5.14: C_d vs. α for Cases S1 and S2.....	147
Figure 5.15: C_m vs. α for Cases S1 and S2.....	148
Figure 5.16: C_p distribution for Case N1 at $\alpha = 9.9^\circ$	149
Figure 5.17: C_p distribution for Case N1 at $\alpha = 14.9^\circ$	149
Figure 5.18: C_l vs. α for the NACA 0012 airfoil at $Re_c = 3 \times 10^6$	150
Figure 5.19: C_d vs. α for the NACA 0012 airfoil at $Re_c = 3 \times 10^6$	151
Figure 5.20: C_m vs. α for the NACA 0012 airfoil at $Re_c = 3 \times 10^6$	152
Figure 5.21: Spanwise C_n and C_l distributions for the NREL UAE Phase VI blade at $V_\infty = 7$ m/s.....	155
Figure 5.22: Spanwise C_n and C_l distributions for the NREL UAE Phase VI blade at $V_\infty = 10$ m/s.....	156
Figure 5.23: Spanwise C_n and C_l distributions for the NREL UAE Phase VI blade at $V_\infty = 25$ m/s.....	157
Figure 5.24: Spanwise C_m distribution for the NREL UAE Phase VI blade at $V_\infty = 7$, $V_\infty = 10$, and $V_\infty = 25$ m/s.	158
Figure B.1: S809 airfoil geometry.....	167
Figure B.2: NACA 0012 airfoil geometry.	170

List of Appendices

Appendix A: Flowchart of the Present Aerodynamic Analysis Algorithm..... 163

Appendix B: Coordinates for the S809 and NACA 0012 Airfoils..... 167

 B.1 S809 Airfoil..... 167

 B.1 NACA 0012 Airfoil 170

Nomenclature

A	Rotor disc area (m^2)
$A_{i,j}$	Influence coefficient of the velocity induced on the control point of panel i by a source sheet located on panel j
a	Wake induction factor, $a = v_i/V_\infty$
B	Turbulent wall-law intercept constant, $B = 5.0$
$B_{i,j}$	Influence coefficient of the velocity induced on the control point of panel i by a vortex sheet located on panel j
C_d	Drag coefficient
C_f	Skin-friction coefficient (Eqn. (2.53))
C_l	Lift coefficient
C_{l_α}	Airfoil lift slope, $C_{l_\alpha} = dC_l/d\alpha$ (rad^{-1})
C_m	Pitching-moment coefficient
C_n	Normal force coefficient
C_p	Rotor power coefficient
C_p	Pressure coefficient
C_Q	Rotor torque coefficient
C_T	Rotor thrust coefficient
C_t	Tangential force coefficient

$C_{x'}$	Force coefficient along the x' axis
$C_{y'}$	Force coefficient along the y' axis
c	Airfoil chord length (m)
D	Drag force per unit span (N/m)
F	Correction factor for Prandtl's root- and tip-loss function; function of Thwaites' pressure gradient
f	Body force per unit mass acting on a fluid element
f_{sep}	Nondimensional chordwise location of flow separation, $f_{sep} = x'_{sep}/c$
H	Shape factor, $H = \delta^*/\theta$
h	Height above ground
h_{ref}	Reference height (usually 10 m above ground)
$\vec{i}$	Unit vector in the x or x' direction
$\vec{j}$	Unit vector in the y or y' direction
K	Linear adverse velocity gradient on the suction surface of the airfoil
k	Roughness length (m)
$\vec{k}$	Unit vector in the z direction
L	Lift force per unit span (N/m); streamwise flow distance on surface from stagnation point to trailing edge (m)
M	Pitching moment per unit span ($N \cdot m/m$)
m	Empirical free-shear-layer angle coefficient
$\dot{m}$	Mass flow rate through rotor disc (kg/s)

N	Turbulent spot production rate parameter (Eqn. (4.54))
N_b	Number of rotor blades
N_{pan}	Number of inviscid flow analysis panels
N_{vis}	Number of viscous flow analysis nodes
n	Turbulent spot production rate (ms^{-1}); rotations per second (s^{-1})
$\bar{\mathbf{n}}$	Unit vector normal to airfoil surface
P	Rotor power (W)
p	Pressure (Pa)
Q	Rotor torque ($\text{N} \cdot \text{m}$)
R	Rotor blade span (m)
Re_c	Reynolds number based on chord length, $\text{Re}_c = V_{\infty} c / \nu$
Re_s	Reynolds number based on streamwise distance, $\text{Re}_s = U_e s / \nu$
Re_θ	Reynolds number based on momentum thickness, $\text{Re}_\theta = U_e \theta / \nu$
Re_{δ^*}	Reynolds number based on displacement thickness, $\text{Re}_{\delta^*} = U_e \delta^* / \nu$
r	Nondimensional radial distance along the blade span; streamwise surface curvature radius (m)
r_0	Nondimensional radial root cut-out distance, $r_0 = y_0 / R$
S	Thwaites' friction factor, $S = \tau_w \theta / \mu U_e$
s	Streamwise distance from leading-edge stagnation point (m)
$\bar{\mathbf{s}}$	Unit vector tangent to airfoil surface

T	Rotor thrust (N)
t	Time (s)
TF	Taylor's turbulence factor, $TF = Tu(L/\lambda_s)^{0.2}$
Tu	Freestream turbulence intensity (%)
U_e	Local freestream velocity at the edge of the boundary layer (m/s)
u	Velocity in the x direction (m/s)
$\vec{V}$	Velocity vector (m/s)
V_∞	Wind velocity (m/s)
V_{∞_r}	Undisturbed freestream velocity relative to a 2D airfoil section or blade element (m/s)
$\vec{V}_{\infty_r}$	Undisturbed freestream velocity vector relative to the 2D airfoil section or blade element (m/s)
v	Velocity in the y direction (m/s)
v_i	Induced velocity (m/s)
$\vec{v}_i$	Induced velocity vector (m/s)
v_w	Velocity deficiency in the wake (m/s)
$\vec{v}_\gamma$	Unit vector of velocity induced by a vortex sheet (m/s)
$\vec{v}_\sigma$	Unit vector of velocity induced by a source sheet (m/s)
w	Velocity in the z direction (m/s)

X Local section speed ratio, $\Omega y/V_\infty$

X_{TSR} Tip-speed ratio, $X_{\text{TSR}} = \Omega R/V_\infty$

x, y, z Cartesian coordinate system (m)

y_0 Radial root cut-out distance (m)

Greek letters

α Angle of attack ($^\circ$); turbulent spot spreading half angle (Eqn. (4.57)) ($^\circ$)

α_0 Zero-lift angle of attack ($^\circ$)

$\alpha_{C_{l_{\max}}}$ Stall angle of attack ($^\circ$)

β Clauser's equilibrium parameter (Eqn. (2.76));

$\beta_{i,j}$ angle between the two lines connecting the control point of panel i to the boundary points of panel j (rad) (Eqn. (4.17))

δ Boundary-layer thickness (m)

δ^* Boundary-layer displacement thickness (Eqn. (2.49)) (m)

ϕ Inflow angle ($^\circ$); potential function; angular position around a cylinder ($^\circ$)

Γ Circulation (m^2/s)

γ Intermittency function; vortex sheet strength

η Nondimensional distance in the boundary-layer transition region

κ von Karman constant, $\kappa = 0.41$

λ	Thwaites' pressure gradient parameter (Eqn. (2.64))
λ_f	Skin friction factor, $\lambda_f = \sqrt{2/C_f}$
λ_i	Inflow ratio, $\lambda_i = (V_\infty + v_i)/\Omega R$
λ_{i_∞}	Freestream inflow ratio, $\lambda_{i_\infty} = V_\infty/\Omega R$
λ_s	Integral length scale of freestream turbulence (m)
μ	Dynamic viscosity (Pa·s)
ν	Kinematic viscosity (m ² /s)
Π	Coles' wake parameter
θ	Boundary-layer momentum thickness (Eqn. (2.51)) (m); local pitch angle of spanwise blade element (°); panel angle with respect to chord line (°)
θ_{α_0}	Local pitch angle of spanwise blade element defined with respect to the zero-lift angle of attack (°)
ρ	Density (kg/m ³)
σ	Local rotor solidity, $\sigma = N_b c/\pi R$; source sheet strength; turbulent spot propagation parameter (Eqn. (4.56))
τ	Shear stress (Pa)
τ_w	Wall shear stress (Pa)
Ω	Rotor angular velocity (rad/s)
ω	Under-relaxation factor
$\vec{\omega}$	Vorticity vector (s ⁻¹)

Overbars

— Time-averaged quantity

Subscripts

IF Inviscid flow

i Current node

$i-1$ Upstream node

$i+1$ Downstream node

i, j Indicates matrix indices

laminar Parameter is calculated by the laminar momentum-integral calculation

m Parameter is evaluated at panel control point

max Maximum

pres Pressure

r Relative to blade element

rot Rotating conditions

sep Parameter is evaluated at the location of boundary-layer separation

skin Skin friction

te Parameter is evaluated at the location of transition completion

ts Parameter is evaluated at the location of transition inception

turbulent Parameter is calculated by the turbulent momentum-integral calculation

VF Viscous flow

x' Parallel to airfoil chord (i.e., in the x' direction)

xy x - y plane

y'	Normal to airfoil chord (i.e., in the y' direction)
γ	Vortex sheet
σ	Source sheet
∞	Undisturbed freestream conditions

Superscripts

'	Coordinate system attached to the airfoil chord; fluctuating quantity
*	Parameter is evaluated at the first turbulent momentum-integral calculation node
n	Normal to the airfoil surface
n	Present iteration
$n-1$	Previous iteration
s	Tangent to the airfoil surface
T	Matrix transpose

Acronyms

2D	Two-dimensional
3D	Three-dimensional
APG	Adverse-pressure-gradient
BEMT	Blade-element momentum theory
BET	Blade-element theory
CFD	Computational fluid dynamics
DNS	Direct numerical simulation

FPG	Favourable-pressure-gradient
HAWT	Horizontal-axis wind turbine
LES	Large eddy simulation
NASA	National Aeronautics Space Administration
NREL	National Renewable Energy Laboratory
RANS	Reynolds-averaged Navier-Stokes
TAT	Thin airfoil theory
T-S	Tollmien-Schlichting
UAE	Unsteady Aerodynamics Experiment
UAV	Unmanned aerial vehicle
ZPG	Zero-pressure-gradient

Chapter 1: Introduction

1.1 Motivation

Understanding and improving the performance of turbomachinery blades has been the focus of many researchers since the beginning of the last century. There is significant room for performance improvements especially in more recent applications such as modern horizontal-axis wind turbines (HAWTs), where complicated problems such as unsteady aerodynamics and the complex structural behaviour of composite blades pose unique challenges to researchers.

When designing and analyzing turbomachinery blades, it is important to consider both the aerodynamic and structural aspects together, since the interaction between the two affects the performance. To this end, the author's research group has set out to provide a complete algorithm to be used for the design optimization of wind turbine and small-scale unmanned aerial vehicle (UAV) propeller blades, modelling both the aerodynamics and structural response of the blade, including their interaction. The present study focuses on the aerodynamic analysis of such blades and since the aerodynamics of wind turbine and propeller blades are fundamentally the same, the algorithm developed in the present study is applicable to both.

If the currently-available methods for analyzing rotating blade aerodynamics are considered, it can be seen that the more flow information a method provides, the greater its computational resource requirements are. For example, direct numerical simulation (DNS) (Section 3.1.4), which numerically integrates the differential equations governing fluid mechanics, provides complete flow information including boundary-layer and separated-flow information at the cost of very high computational requirements. On the

other hand, the blade-element-momentum theory, which analyzes the blade by dividing it into a finite number of spanwise elements, requires only the angle of attack of each blade element to determine the aerodynamic forces with no viscous flow information, at a very low computational cost. There are methods in between these two extremes, such as Reynolds-averaged Navier-Stokes methods (Section 3.1.4), which numerically solve the time-averaged forms of the differential equations governing fluid mechanics; and viscous-inviscid interaction methods (Section 3.1.5), which treat regions of the flowfield as either viscous or inviscid, and use an appropriate method to analyze the flow in each region. The present study uses a viscous-inviscid interaction method as a starting point and aims to update this method based on the latest understanding of flow physics related to airfoils.

1.2 Objectives

The aim of the present study is to provide a complete algorithm for the aerodynamic analysis of rotating blades for configurations where the aerodynamics of each blade can be treated in isolation, by developing an algorithm around a weak viscous-inviscid interaction method (Section 3.1.5) used to calculate the aerodynamics of airfoil geometries. The study also aims to update this weak viscous-inviscid interaction method by making use of the latest models and correlations to predict specific flow phenomena such as transitioning flow and separated flow. The complete algorithm is to provide sufficiently-accurate predictions for unstalled and unyawed operating conditions, have a modest computational power requirement, provide as much flow information as possible,

and be able to identify if the blade is approaching the limits of its optimal performance zone.

1.3 Approach

The present study combines a number of well-established methods, one for each step of the aerodynamic analysis, in one single algorithm. The rotating blade is analysed by dividing it into a finite number of spanwise elements, and the angle of attack and Reynolds number of each blade element are calculated by using the blade-element-momentum theory with root- and tip-loss corrections. Once the operating conditions of the blade element are calculated, the aerodynamic performance of each blade element is calculated using a two-dimensional (2D) weak viscous-inviscid interaction method. Finally, corrections to the predicted 2D aerodynamic coefficients to account for radial (spanwise) flow effects due to the rotation of the blade are applied by using so-called “stall delay” models. The algorithm has been coded in MATLAB® to benefit from its vast library of pre-defined functions and de-bugging tools, which greatly simplify the programming effort.

Chapters 2 and 3 of this manuscript provide information on the concepts and methods relevant to rotating-blade aerodynamics and viscous-inviscid interaction. Readers who are experienced in these fields may wish to advance to Chapters 4 and 5, in which the numerical methods and prediction capabilities of the present algorithm are discussed in detail, respectively.

Chapter 2: Background Information on Rotating Blade Aerodynamics

In this chapter, brief background information on concepts and methods that form the basis of the present algorithm is presented, with the focus being on the low-speed aerodynamics of wind turbine and propeller blades. All subjects covered in this chapter can be found in relevant fluid mechanics and aerodynamics textbooks, such as the works of Anderson (2011), Katz & Plotkin (1991), Leishman (2006), and White (2006; 2011). They are presented here to provide a background for the reader who is not experienced in the field.

After a brief overview of the aerodynamic working principle of wind turbine and propeller blades, basic methods to analyze the performance of such turbomachinery are presented from a wind turbine perspective. Next, the governing equations of fluid mechanics are presented, followed by the simplifications applied to these equations for inviscid flow and boundary-layer analyses, along with some additional background information on boundary layers.

2.1 Aerodynamic Working Principle of Wind Turbine and Propeller Blades

Wind turbines extract energy from the wind by using lifting bodies (blades) to create lift force. The lift force is used to create rotation and by the use of an electric generator, the mechanical rotation energy is converted to electrical energy. Propellers do the opposite; they impart energy to a fluid by utilizing a combustion engine or electric motor to rotate the propeller blades creating lift and thrust. From the aerodynamics point of view, it is important to know how the lift force is generated by the blades and how

drag effects are countered. In Figure 2.1, a section of a blade is shown with the associated flow velocities and the resulting aerodynamic forces, in the case of a wind turbine.

Figure 2.1: Velocities and aerodynamic forces on a wind turbine blade. Reproduced from Hau (2006).

The resultant velocity relative to the airfoil, $V_{\infty,r}$, is the sum of the axial velocity and the rotational velocity, Ωy , where Ω is the angular velocity of the rotor and y is the distance of the blade cross section from the axis of rotation. The angle between the resultant velocity and the chord line of the airfoil is called the angle of attack, α . The resultant forces acting on the airfoil are usually expressed as two perpendicular forces (lift and drag) and a pitching moment at the aerodynamic centre of the airfoil, which is defined as the location where the pitching-moment coefficient does not vary with the angle of attack, located at 25% of the chord length from the leading edge for subsonic

airfoils. The generated lift and drag forces are perpendicular and parallel to the relative velocity, V_{∞} , respectively. The sum of lift and drag forces is the resultant net force produced by the blade segment, and its axial component is the thrust force generated by the rotating blade. In wind turbines, the resultant force component in the plane of rotation generates the torque, which enables the generator to produce electricity. In practice, the lift and drag forces and the pitching moment are generally normalized into coefficient form:

$$C_l = \frac{L}{\frac{1}{2} \rho V_{\infty}^2 c} \quad (2.1)$$

$$C_d = \frac{D}{\frac{1}{2} \rho V_{\infty}^2 c} \quad (2.2)$$

$$C_m = \frac{M}{\frac{1}{2} \rho V_{\infty}^2 c^2} \quad (2.3)$$

where C_l , C_d , and C_m are respectively the lift, drag, and pitching-moment coefficients; L , D , and M are respectively the lift and drag forces and the pitching moment per unit blade span; ρ is the density of the fluid; and c is the chord length of the airfoil.

2.2 Methods for Calculating Turbomachinery Performance

In this section, some of the well-established methods for the overall aerodynamic and performance analyses of turbomachinery are reviewed. These methods are the momentum (actuator disc) theory, blade-element theory, and the blade-element-momentum theory.

2.2.1 Momentum (Actuator Disc) Theory

Figure 2.2: Momentum theory model of a wind turbine. Reproduced from Leishman (2006).

The momentum theory assumes a one-dimensional control volume bounded by the surface of the stream tube and two cross sections of the stream tube, as seen in Figure 2.2. The same theory shown here for turbines also applies to propellers, which have the same underlying physics, with only two mathematical differences. The first difference is that in turbines, energy is extracted from the fluid (i.e., the fluid is doing work on the turbine) whereas the opposite is true for propellers. A stream tube going through a turbine expands since kinetic energy is extracted from it, while a stream tube going through a propeller contracts. The second difference is the flow direction being opposite for propellers from the direction shown in Figure 2.2, while keeping the stream tube in the same orientation shown in the figure. This difference also causes the direction of the induced velocity at the propeller disc plane to be in the same direction as the

freestream, since the induced velocity, v_i , does not change direction between the two applications.

The assumptions that are made for the development of the theory are listed by Manwell et al. (2009) as follows:

- homogenous, incompressible, steady-state flow,
- no friction drag,
- infinite number of blades,
- uniform thrust distribution over the disc or rotor area,
- a non-rotating wake, and
- undisturbed ambient static pressure far upstream and downstream of the rotor.

Referring to Figure 2.2, Leishman (2006) explains the theory as follows. The approaching wind speed is V_∞ , with the induced velocity in the plane of the turbine blades being v_i . The net velocity at the plane of the turbine is thus $V_\infty - v_i$. Because the turbine extracts energy from the wind, thus reducing its static pressure immediately downstream of the rotor, velocity far downstream of the turbine will be lower than upstream of the turbine; in propellers, the behaviour will be opposite, hence the flow direction will be the opposite of that shown in Figure 2.2. Thus, well downstream of the turbine, in the far wake, the velocity is $V_\infty - v_w$, where v_w is the velocity deficiency in the wake. Also, to satisfy continuity conditions under an incompressible flow assumption, the wake downstream of the turbine must expand, since the flow velocity downstream is lower. For propellers, the wake contracts because the flow velocity downstream is higher.

The mass flow rate through the turbine disc, $\dot{m}$, is:

$$\dot{m} = \rho A (V_\infty - v_i) \quad (2.4)$$

where ρ is the air density and A is the cross-sectional area of the turbine rotor. Given that the static pressures far upstream and downstream of the turbine will match the ambient pressure, the thrust, T , is related to the change of momentum across the actuator turbine disc:

$$T = \dot{m}V_\infty - \dot{m}(V_\infty - v_w) = \dot{m}v_w \quad (2.5)$$

The work done by the air on the turbine per unit time, P , is:

$$P = \frac{1}{2} \dot{m}V_\infty^2 - \frac{1}{2} \dot{m}(V_\infty - v_w)^2 = \frac{1}{2} \dot{m}v_w (2V_\infty - v_w) \quad (2.6)$$

This power output of the turbine can also be expressed as:

$$P = T(V_\infty - v_i) = \dot{m}v_w (V_\infty - v_i) \quad (2.7)$$

Comparing the power expressions in Eqns. (2.6) and (2.7), it is evident that

$$v_w = 2v_i \quad (2.8)$$

Expanding Eqn. (2.7) using Eqns. (2.4) and (2.8) gives:

$$P = 2\rho A (V_\infty - v_i)^2 v_i \quad (2.9)$$

The power coefficient, C_p , which is typically used to express rotor performance, is defined as:

$$C_p = \frac{P}{\frac{1}{2} \rho A V_\infty^3} = \frac{\text{Rotor power}}{\text{Power in the wind}} \quad (2.10)$$

for wind turbines. Substituting Eqn. (2.9) into Eqn. (2.10) gives:

$$C_p = \frac{2\rho A (V_\infty - v_i)^2 v_i}{\frac{1}{2} \rho A V_\infty^3} = \frac{4(V_\infty - v_i)^2 v_i}{V_\infty^3} = 4a(1-a)^2 \quad (2.11)$$

where $a = v_i/V_\infty$ is the fractional decrease in wind velocity between the freestream and the rotor plane, called the *induction factor*.

The thrust coefficient for wind turbines is defined as:

$$C_T = \frac{T}{\frac{1}{2} \rho A V_\infty^2} \quad (2.12)$$

The thrust acting on the turbine is calculated by substituting Eqn. (2.4) into Eqn. (2.5):

$$T = \dot{m} v_w = \rho A (V_\infty - v_i) v_w = 2 \rho A (V_\infty - v_i) v_i \quad (2.13)$$

Using this thrust expression in the thrust coefficient yields:

$$C_T = \frac{2 \rho A (V_\infty - v_i) v_i}{\frac{1}{2} \rho A V_\infty^2} = \frac{4 (V_\infty - v_i) v_i}{V_\infty^2} = 4a(1-a) \quad (2.14)$$

The induction factor for the maximum power coefficient, $C_{P_{max}}$, is determined by taking the derivative of Eqn. (2.11) with respect to a and setting it equal to zero. This results in $a = 1/3$. This induction ratio value corresponds to the maximum C_P and C_T values:

$$C_{P_{max}} = \frac{16}{27} = 0.593 \quad (2.15)$$

$$C_{T_{max}} = \frac{8}{9} = 0.889 \quad (2.16)$$

These maximum values correspond to the Betz-Lanchester limit (Glauert, 1935; 1983), which are the maximum theoretically-possible power and thrust coefficients. In reality, three effects decrease the maximum achievable power coefficient. As listed by Manwell et al. (2009), these effects are:

- rotation of the wake downstream of the rotor,

- a finite number of blades and associated root and tip losses, and
- aerodynamic drag.

Since $V - v_w > 0$, for the flow direction to remain as shown in Figure 2.2, the momentum theory is only valid for $0 \leq a \leq 0.5$. A turbine operating within this range is in the *windmill-brake state*, outside of which the momentum theory breaks down. As Leishman (2006) explains, for $0.5 < a \leq 1$, the turbine is in what is called the *turbulent-wake state*. Close to $a = 1$, the rotor enters the *vortex-ring state*, in which the blade-tip vortices, which resemble helically-wound vortex filaments trailing the tips of the blades, approach the blade tips and cause significant unsteady aerodynamic effects on the flow passing through the rotor plane. To determine the induction factor when the turbine is in the turbulent-wake or vortex-ring states, an empirical correlation between the thrust coefficient and the induction factor is used. This empirical correlation is based on the data provided by Lock (1925), and it was first suggested by Glauert (1926) with later modifications by Burton et al. (2001), Buhl (2005), and Leishman (2006). Leishman (2006) suggests using:

$$C_T = 4a(a-1) + 2 \quad (2.17)$$

To be able to use the momentum theory, the induction factor, a , needs to be known *a priori*. In addition, the many assumptions and simplifications made in the process of developing the theory cause its results to be trusted only as a first-order prediction (Leishman, 2006).

2.2.2 Blade-Element Theory (BET)

In this section, the blade-element theory (BET) is presented from the perspective of wind-turbine blade design, as described by Leishman (2006). This method is based on dividing the lift-generating body into a finite number of segments (elements), assuming each element behaves as a two-dimensional (2D) airfoil section. The calculated power and thrust at each blade element is then integrated over the blade span to calculate the performance of the full blade.

Figure 2.3: Blade-element model for a wind turbine operating in axial (unyawed) flow. Reproduced from Leishman (2006).

As shown in Figure 2.3, the axial flow is directed toward the plane of rotation of the blades. θ is the local pitch angle of the blade element and the angle of attack of the blade element is therefore $\alpha = \theta + \phi$, in which ϕ is the inflow angle relative to the plane of rotation. The induced inflow, v_i , changes the direction of the relative flow velocity and alters the angle of attack at each blade element. The resultant velocity acting on the blade element is V_{∞} . The forward inclination of the lift force, dL , allows the component of the lift in the plane of rotation to overcome the corresponding in-plane component of the drag force, dD , thus resulting in a net in-plane force that drives the wind turbine to produce power.

To derive the thrust and power coefficients, lift and drag forces acting on the blade element are resolved perpendicular and parallel to the rotor disc plane, and their contributions to thrust, torque, and power are calculated. The full derivation can be found in rotor aerodynamics textbooks such as the work of Leishman (2006). Before presenting the final equations to calculate the thrust and power coefficients, the primary terms that appear in those equations are presented.

$r = y/R$ is the nondimensional radial distance, where y is the radial distance of the blade element from the rotation axis and R is the blade span.

$\sigma = N_b c / \pi R$ is called the *local* or *chord solidity* of the rotor, where N_b is the number of blades comprising the turbine rotor and c is the local blade chord length.

$X = \Omega y / V_{\infty}$ is called the *local section speed ratio*, where Ω is the angular velocity of the turbine blade and V_{∞} is the velocity perpendicular to the plane of rotation, i.e., the wind speed (assuming the turbine is not yawed).

$X_{\text{TSR}} = \Omega R / V_{\infty}$, which is called the *tip-speed ratio*, is an important characteristic of wind turbines, and it is the ratio of the in-plane tip speed to the wind speed. The following equation shows the relationship between X and X_{TSR} .

$$X = \frac{\Omega y}{V_{\infty}} = \frac{\Omega R}{V_{\infty}} \left(\frac{\Omega y}{\Omega R} \right) = X_{\text{TSR}} r \quad (2.18)$$

Using these parameters, the incremental thrust and power coefficients for one blade element are expressed respectively as,

$$dC_T = \sigma X_{\text{TSR}}^2 (C_l + \phi C_d) r^2 dr \quad (2.19)$$

$$dC_p = \sigma X_{\text{TSR}}^3 (\phi C_l - C_d) r^3 dr \quad (2.20)$$

where C_l and C_d are respectively the lift and drag coefficients of the 2D airfoil at the local angle of attack, α . To calculate C_T and C_p for the full blade (or turbine), the incremental thrust and power coefficients dC_T and dC_p are integrated along the span of each blade. However, to evaluate these coefficients, the induced inflow velocity, v_i , and the sectional aerodynamic coefficients of the airfoil (C_l and C_d) are needed. In this case, the fundamental problem is to determine the induced inflow velocity. Leishman (2006) indicates that it is possible to proceed with the calculations using closed-form analytical solutions based on certain assumptions and approximations that deviate from the three-dimensional (3D) nature of the problem, such as assuming untwisted blades with uniform inflow. Instead, the blade-element-momentum theory may be used to obtain a more accurate prediction.

2.2.3 Blade-Element-Momentum Theory (BEMT)

The BEMT is a hybrid method, developed by combining the momentum (actuator disc) and blade-element theories. The BEMT is briefly explained here from a wind turbine perspective, as described by Leishman (2006). As discussed in the momentum theory, the incremental thrust on an annulus of the turbine disc is calculated using the momentum change across the disc annulus. The mass flow rate through an annulus of the disc is (referring to Figure 2.3):

$$d\dot{m} = \rho dA (V_\infty - v_i) = 2\pi \rho (V_\infty - v_i) y dy \quad (2.21)$$

The incremental thrust on the annulus, from Eqn. (2.13), is:

$$dT = 2\rho (V_\infty - v_i) v_i dA = 4\pi \rho (V_\infty - v_i) v_i y dy \quad (2.22)$$

If Eqn. (2.22) is rewritten in the form of a thrust coefficient with $r = y/R$:

$$dC_T = 8 \left(1 - \frac{v_i}{V_\infty} \right) \frac{v_i}{V_\infty} r dr = 8a(1-a)r dr \quad (2.23)$$

If a is assumed to be constant over the disc area, Eqn. (2.23) integrates to the result for C_T given by the momentum theory in Eqn. (2.14).

The key step in the blade-element-momentum theory is to equate the thrust from the momentum and blade-element theories, given by Eqns. (2.23) and (2.19) respectively:

$$dC_T = 8a(1-a)r dr = \sigma X_{\text{TSR}}^2 (C_l + \phi C_d) r^2 dr \quad (2.24)$$

The next step requires relating C_l and C_d to the angle of attack, α , of the blade element.

Here, a simplification is used, which recognizes that C_l/C_d is generally large and the product ϕC_d is small. With this simplification, Eqn. (2.24) is written as:

$$dC_T = \sigma X_{\text{TSR}}^2 C_l r^2 dr \quad (2.25)$$

If it is assumed that the blade is not stalled, and the blade does not operate very close to its peak lift coefficient value, the relationship between C_l and α is linear (Leishman, 2006). Therefore, C_l can be expressed in terms of the slope, C_{l_α} , of the linear relation between C_l and α :

$$C_l = C_{l_\alpha} (\theta - \alpha_0 + \phi) \quad (2.26)$$

where $\phi = \left(\frac{1-a}{rX_{\text{TSR}}} \right)$, and α_0 is the zero-lift angle of attack for the airfoil section.

Accordingly,

$$C_l = C_{l_\alpha} \left(\theta - \alpha_0 + \frac{1-a}{rX_{\text{TSR}}} \right) \quad (2.27)$$

If θ is defined with respect to the zero-lift angle, α_0 , the result is:

$$\theta_{\alpha_0} = \theta - \alpha_0 \quad (2.28)$$

Equating Eqns. (2.23) and (2.25), and expanding C_l as per Eqns. (2.27) and (2.28) gives:

$$8a(1-a) = \sigma X_{\text{TSR}}^2 C_l r = \sigma X_{\text{TSR}} C_{l_\alpha} (X_{\text{TSR}} \theta_{\alpha_0} r + (1-a)) \quad (2.29)$$

Rearranging Eqn. (2.29) yields:

$$a^2 - \left(\frac{\sigma X_{\text{TSR}} C_{l_\alpha}}{8} + 1 \right) a + \frac{\sigma X_{\text{TSR}} C_{l_\alpha} (X_{\text{TSR}} \theta_{\alpha_0} r + 1)}{8} = 0 \quad (2.30)$$

If this equation is solved for the unknown induction factor a , the result is:

$$a(r, X_{\text{TSR}}) = -\sqrt{\left(\frac{\sigma X_{\text{TSR}} C_{l_\alpha}}{16} + \frac{1}{2} \right)^2 - \frac{\sigma X_{\text{TSR}} C_{l_\alpha} (X_{\text{TSR}} \theta_{\alpha_0} r + 1)}{8}} + \left(\frac{\sigma X_{\text{TSR}} C_{l_\alpha}}{16} + \frac{1}{2} \right) \quad (2.31)$$

For the BEMT as it applies to wind turbines, (2.31) is the fundamental equation. As an initial estimation, $C_{l_\alpha} = 2\pi$ may be used. Alternatively, if 2D airfoil data is available, the

actual value of the lift slope may be used directly. For any given blade pitch, spanwise blade twist, chord length, and airfoil section distribution, Eqn. (2.31) allows the induction factor, a (thus the induced velocity v_i), to be calculated for any blade element. The only requirement, other than the blade geometry and the operating condition (X_{TSR}), is the 2D lift slope of the airfoil of the blade element. However, Eqn. (2.31) is only valid for the range $0 \leq a \leq 0.5$, since this is also the limitation of the underlying momentum theory. For $a > 0.5$, the fundamental equation for the BEMT is rederived by equating the momentum (Eqn. (2.17)) and blade-element (Eqn. (2.19)) results of an infinitesimally-small blade element operating in this state (Leishman, 2006), giving:

$$8a(a-1)+4=\sigma X_{TSR}^2 C_{l_\alpha} r = \sigma X_{TSR} C_{l_\alpha} (X_{TSR} \theta_{\alpha_0} r + (1-a)) \quad (2.32)$$

Rearranging Eqn. (2.32) yields:

$$a^2 + \left(\frac{\sigma X_{TSR} C_{l_\alpha}}{8} - 1 \right) a + \left(\frac{1}{2} - \frac{\sigma X_{TSR} C_{l_\alpha} (X_{TSR} \theta_{\alpha_0} r + 1)}{8} \right) = 0 \quad (2.33)$$

which has the solution:

$$a(r, X_{TSR}) = \sqrt{\left(\frac{\sigma X_{TSR} C_{l_\alpha}}{16} - \frac{1}{2} \right)^2 - \left(\frac{1}{2} - \frac{\sigma X_{TSR} C_{l_\alpha} (X_{TSR} \theta_{\alpha_0} r + 1)}{8} \right)} - \left(\frac{\sigma X_{TSR} C_{l_\alpha}}{16} - \frac{1}{2} \right) \quad (2.34)$$

Two important points of interest are observed in Eqn. (2.31). First, for a given tip-speed ratio, the twist distribution that yields a uniform relative inflow (uniform value of the induction factor) is achieved if $\theta r = \text{constant}$, which is known as *ideal twist*. This twist distribution, which can also be expressed as $\theta(r) = \theta_{ip}/r$, yields the most efficient energy extraction for a wind turbine, since it gives the lowest induced losses. Second, in

the development of the momentum theory, it is shown that $a=1/3$ for maximum efficiency. This value of a allows Eqn. (2.31) to be solved for the blade pitch necessary to achieve the most efficient operation at a given tip-speed ratio for a wind turbine with variable-pitch blades.

2.3 Governing Equations of Fluid Dynamics

The fundamental governing equations of fluid dynamics are second-order nonlinear partial differential equations that are based on the principles of the conservation of mass, momentum, and energy within a fluid. Since the flow is incompressible and steady in the present study, the equation for energy conservation is not presented. For a Newtonian fluid (fluid in which the shear stress varies linearly with strain rate, i.e., fluid viscosity is independent of strain rate) in a non-accelerating Cartesian coordinate system, the relevant mass and momentum conservation equations for steady flow are:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (\text{Mass conservation}) \quad (2.35)$$

$$\rho \left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = -\frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + \rho f_x \quad (\text{x-momentum conservation}) \quad (2.36)$$

$$\rho \left(u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = -\frac{\partial p}{\partial y} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + \rho f_y \quad (\text{y-momentum conservation}) \quad (2.37)$$

$$\rho \left(u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = -\frac{\partial p}{\partial z} + \mu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + \rho f_z \quad (\text{z-momentum conservation}) \quad (2.38)$$

where ρ is the density of the fluid; u, v , and w are respectively the velocity components in the x, y , and z Cartesian directions; p is the pressure; μ is the dynamic viscosity; and f_i is the body force (e.g., gravity) per unit mass acting on a fluid element. The momentum conservation equations are known as the Navier-Stokes equations. In this manuscript, the momentum and mass conservation equations are collectively referred to as the Navier-Stokes equations, as is common practice in published literature.

These equations have only a limited number of known analytical solutions; however, they can be solved numerically by computational fluid dynamics (CFD) algorithms (White, 2011). Since the Navier-Stokes equations are inherently difficult to solve analytically, and computationally expensive to evaluate, many simplified methods have been developed for various fluid dynamics applications (White, 2011). One such simplification is to assume that the flow is inviscid, meaning that the effect of viscosity on the flow can be fully neglected. This simplification may be applied to regions of fluid flow with negligible viscous forces.

2.4 Inviscid Flow

2.4.1 Euler Equations

If the flow is assumed to be inviscid, the viscous terms in the Navier-Stokes equations are dropped, resulting in a set of equations called the Euler equations. The Euler equations can be evaluated numerically to provide significant insight into complicated flows for which the assumption of inviscid flow is acceptable (Leishman, 2006).

2.4.2 Potential Flow Equations

If a further assumption of irrotational flow (zero vorticity) in addition to inviscid flow is made, the governing equations can be further simplified. If the flow is irrotational ($\vec{\omega} = \nabla \times \vec{V} = 0$, where $\vec{\omega}$ is the vorticity vector, and $\vec{V} = u\vec{i} + v\vec{j} + w\vec{k}$, with $\vec{i}$, $\vec{j}$, and $\vec{k}$ being the unit vectors in the x , y , and z directions, respectively), the velocity field can be expressed as the gradient of a potential function ϕ , as:

$$\vec{V} = \nabla \phi \quad (2.39)$$

so that

$$u = \frac{\partial \phi}{\partial x} \quad v = \frac{\partial \phi}{\partial y} \quad w = \frac{\partial \phi}{\partial z} \quad (2.40)$$

The potential equation for a steady, incompressible flow (Laplace's equation) is:

$$\nabla^2 \phi = \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0 \quad (2.41)$$

If the potential function $\phi(x, y, z)$ is known for the flow, the velocity field can be readily calculated. However, this approach is not recommended for flows where rotationality is important, such as wake flow (Leishman, 2006).

Partial differential equations such as Eqn. (2.41) can be numerically evaluated relatively easily by using finite difference, finite element, or finite volume methods (White, 2006). Each of these methods requires the flowfield to be divided into a mesh in order to evaluate the necessary functions at predefined locations. Finite difference methods approximate partial derivatives by using algebraic differences between discrete mesh points (*nodes*) (White, 2006), whereas finite element methods model the functions by approximating them as algebraic combinations of local nodal values in order to

evaluate their values between the mesh points (White, 2011). Finite volume methods model the flowfield as discrete cells (with finite volume) and evaluate surface integral forms of the governing equations over each cell surface in order to calculate the flux of the target variables between adjacent cells (Leishman, 2006). However, the necessary computation time for these methods makes them unsuitable to be used for design optimization. Therefore, such methods are not preferable for the purposes of the present study.

2.4.3 Potential-Flow-Singularity Methods

Instead of solving Laplace's equation (Eqn. (2.41)), potential flow singularities (e.g., sources/sinks, vortices, and doublets) of unknown strength placed at strategic locations (e.g., chord line of an airfoil, surface of a body) within the flowfield may be used to model inviscid flow (Anderson, 2011). Each singularity placed inside the flowfield modifies the velocity field in a specific way and the combined effect of all singularities on the flowfield is calculated in order to find the necessary strength values that satisfy certain boundary conditions (e.g., no flow going through solid surfaces). These methods provide much faster solutions to inviscid flow problems, since they usually involve solving a set of linear algebraic equations instead of partial differential equations. Such methods include the thin airfoil theory, lifting-line blade model, and panel methods, which are all discussed in Chapter 3. In addition, a simple panel method that is used in the present algorithm is further detailed in Chapter 4.

2.5 Viscous Flow

Simplifications to the Navier-Stokes equations can also be made for flows where the effects of viscosity cannot be neglected. It is most important to take viscous effects into account when analyzing regions of flow close to solid boundaries of fluid flows, called the *boundary layer*. In what follows, simplifications that are applicable to the Navier-Stokes equations for a boundary-layer analysis are explained and some background information on boundary layers is presented.

2.5.1 Boundary-Layer Equations

Since a two-dimensional, steady, incompressible, viscous flow most closely corresponds to the flow problem examined in the present study, the Navier-Stokes equations are reduced to a continuity equation and two momentum equations along the x and y directions, which are parallel to, and normal to the wall, respectively (White, 2006). Terms for the body forces are also neglected, since the buoyancy of the working fluid (air) is not dominant. The resulting set of equations is:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (\text{continuity}) \quad (2.42)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (\text{x-momentum}) \quad (2.43)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (\text{y-momentum}) \quad (2.44)$$

where ν is the kinematic viscosity of the fluid, and u and v are velocity components in the x and y directions, respectively.

Prandtl (1905) showed that if the Reynolds number is greater than 1000, the boundary layer must be very thin, such that the gradients of the fluid properties in the wall normal (y) direction are much greater than the gradients along the wall direction (x). Applying this fact to Eqn. (2.44) in the absence of significant streamwise wall surface curvature, leads to:

$$\frac{\partial p}{\partial y} \approx 0 \quad (2.45)$$

The edge of the boundary layer ($y = \delta$, where δ is the boundary-layer thickness) is defined as the location where the local velocity is equal to 99% of the local freestream velocity, $U_e(x)$. From Eqn. (2.45), the pressure at the wall is assumed to be the same as the pressure at the boundary-layer edge, so that the momentum equation in the freestream above $y = \delta$ for a steady flow becomes:

$$U_e \frac{dU_e}{dx} = -\frac{1}{\rho} \frac{dp}{dx} \quad (2.46)$$

Using Eqn. (2.46), the pressure term is eliminated from the x -momentum equation in the boundary layer. After these simplifications, the resulting set of boundary-layer equations is:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (\text{continuity}) \quad (2.47)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = U_e \frac{dU_e}{dx} + \nu \frac{\partial^2 u}{\partial y^2} \quad (\text{streamwise momentum balance}) \quad (2.48)$$

There are two important integral quantities relevant to boundary-layer flows; the *displacement thickness* (δ^*) and the *momentum thickness* (θ). The displacement

thickness is the amount of deflection of the inviscid flow streamlines away from the wall due to mass conservation, since the flow in the boundary layer is slowed down by viscous drag (White, 2006). For incompressible flows, it is defined as:

$$\delta^* = \int_0^\delta \left(1 - \frac{u}{U_e}\right) dy \quad (2.49)$$

The *momentum thickness* is a measure of the momentum loss, or *skin-friction drag*, in the boundary layer, due to the no-slip condition at the wall. As White (2006) explains, the momentum loss, or skin friction in the boundary layer is expressed as:

$$D_{skin} = \rho \int_0^\delta u(U_e - u) dy \quad (2.50)$$

By definition, the momentum thickness for incompressible flows is:

$$\theta = \frac{D_{skin}}{\rho U_e^2} = \int_0^\delta \frac{u}{U_e} \left(1 - \frac{u}{U_e}\right) dy \quad (2.51)$$

The ratio of the displacement thickness and the momentum thickness, $H = \delta^*/\theta$, is called the *shape factor*. The shape factor value varies from ~2.0 to ~3.5 for laminar boundary layers and from ~1.3 to ~2.5 for turbulent boundary layers; a lower shape factor indicates a fuller velocity profile (White, 2006; 2011).

The wall shear stress per unit width, τ_w , is used to calculate the total drag caused by the skin friction (i.e., the momentum loss in the boundary layer) by integration along the wall:

$$D_{skin} = \int_0^L \tau_w(x) dx \quad (2.52)$$

where L is the total flow distance along the surface. The skin-friction drag can be normalized into a coefficient form:

$$C_f = \frac{\tau_w(x)}{\frac{1}{2}\rho U_e^2} \quad (2.53)$$

where C_f is called the *skin-friction coefficient*. The corresponding normalized drag force is:

$$C_{d_{skin}} = \frac{D_{skin}}{\frac{1}{2}\rho V_\infty^2 L} \quad (2.54)$$

which is the *skin-friction drag coefficient* per unit surface width, where V_∞ is the undisturbed freestream velocity approaching the surface.

It is also very important to note that the boundary-layer equations are only valid up to the streamwise location of flow separation, due to the Goldstein singularity (Section 3.1.5) at the separation point (Goldstein, 1948). As White (2006) explains, on continuous wall surfaces, flow separation occurs under adverse streamwise pressure gradients ($dp/dx > 0$ or $dU_e/dx < 0$). The freestream deceleration causes the wall shear to decrease, and where the wall shear reaches zero, the boundary layer separates from the surface. Separated flow is arguably the most difficult type of flow to model; however, it is a common occurrence in flows over lifting bodies at moderate and high angles of attack (Katz & Plotkin, 1991).

As explained by White (2006), the boundary-layer equations that are presented up to this point are valid for the instantaneous analysis of both laminar and turbulent flows. However, turbulent flows have historically been analyzed as having two separate components due to the significant velocity fluctuations within the flow; a mean-flow (time-averaged) component, and a fluctuating component. If the Navier-Stokes equations are rederived using the two components, the Reynolds-averaged Navier-Stokes (RANS)

equations are obtained. If the two-dimensional, incompressible turbulent-boundary-layer equations are derived using the RANS equations, the result is:

$$\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} = 0 \quad (\text{continuity}) \quad (2.55)$$

$$\bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} = U_e \frac{dU_e}{dx} + \nu \frac{\partial^2 \bar{u}}{\partial y^2} - \frac{\partial \overline{u'v'}}{\partial y} \quad (\text{streamwise momentum balance}) \quad (2.56)$$

in which the terms with an overbar indicate time-averaged flow components, and the terms with a prime symbol indicate fluctuation components, so that the instantaneous velocities in the x and y directions are expressed as:

$$u = \bar{u} + u' \quad (2.57)$$

$$v = \bar{v} + v' \quad (2.58)$$

respectively. The addition of the fluctuation terms as new unknowns complicates the solution and makes the numerical evaluation of the boundary-layer equations computationally demanding. Therefore, another simpler and computationally-efficient method for solving the boundary-layer equations is necessary for the purposes of the present study. White (2006) suggests that the instantaneous boundary-layer equations can be integrated from the wall to infinity to get an ordinary differential equation for the boundary layer that is valid for both laminar and turbulent flows. The most commonly used such equation is the momentum-integral relation (White, 2006).

2.5.2 The Momentum-Integral Relation

The momentum-integral relation is an approximate integral equation that is used to analyze boundary-layer flows with pressure gradients, which is typical of flows over an airfoil. As White (2006) explains, to derive the momentum-integral relation for a

laminar boundary layer, the continuity equation (Eqn. (2.47)) is multiplied by $(u - U_e)$ and subtracted from the streamwise momentum equation of the boundary layer (Eqn. (2.48)):

$$-\frac{1}{\rho} \frac{\partial \tau_{xy}}{\partial y} = \frac{\partial}{\partial x} (uU_e - u^2) + (U_e - u) \frac{dU_e}{dx} + \frac{\partial}{\partial y} (vU_e - vu) \quad (2.59)$$

where ρ is the fluid density, τ_{xy} is the shear stress, U_e is the velocity at the edge of the boundary layer, and u and v are the velocities in the x and y directions, respectively. Eqn. (2.59) is integrated from the wall to the edge of the boundary layer. The result of the integration is:

$$\frac{\tau_w}{\rho} = \frac{\partial}{\partial x} \int_0^\delta u(U_e - u) dy + \frac{dU_e}{dx} \int_0^\delta (U_e - u) dy \quad (2.60)$$

where τ_w is the shear stress at the wall. The integrals of $(U_e - u)$ and $u(U_e - u)$ relate to the displacement and momentum thicknesses, respectively, given in Eqns. (2.49) and (2.51). Rewriting Eqn. (2.60) with the integral thicknesses leads to:

$$\frac{\tau_w}{\rho U_e^2} = \frac{C_f}{2} = \frac{d\theta}{dx} + (2\theta + \delta^*) \frac{1}{U_e} \frac{dU_e}{dx} \quad (2.61)$$

where C_f is the skin-friction coefficient, θ is the momentum thickness, and δ^* is the displacement thickness. This equation can be rewritten based on the shape factor, H , as:

$$\frac{C_f}{2} = \frac{d\theta}{dx} + (2 + H) \frac{\theta}{U_e} \frac{dU_e}{dx} \quad (2.62)$$

where $H = \delta^* / \theta$. A similar derivation for the turbulent boundary layer yields:

$$\frac{C_f}{2} = \frac{d\theta}{dx} + (2 + H) \frac{\theta}{U_e} \frac{dU_e}{dx} + \frac{1}{U_e^2} \int_0^\delta \frac{d}{dx} (\overline{u'^2} - \overline{v'^2}) dy \quad (2.63)$$

where $\overline{u'^2}$ and $\overline{v'^2}$ are the time-averaged values of the squares of the x and y components of fluctuating velocities, respectively. The additional integral term appearing in the turbulent momentum-integral equation is typically negligibly small except near separation (Abu-Ghannam & Shaw, 1980).

The momentum-integral relation has three unknown boundary-layer parameters: θ , C_f , and H ; therefore two additional closure equations are needed to solve the system. The necessary closure equations are provided by different methods for laminar and turbulent flow. These are presented in Sections 2.5.2.1 and 2.5.2.2, respectively. The solution of the resultant sets of equations for the streamwise development of θ , C_f , and H is presented in Chapter 4.

2.5.2.1 Laminar Closure Method

For laminar flows, the unknown boundary-layer parameters are closely related to the velocity profile, which can be approximated by using a single parameter, providing an easy way to close the equations (White, 2006). Thwaites (1949) has suggested using empirical correlations to provide closure to the momentum-integral equation in laminar boundary layers, by rewriting the momentum-integral equation (Eqn. (2.62)) in terms of a pressure gradient parameter, λ , defined as:

$$\lambda = \frac{\theta^2}{\nu} \frac{dU_e}{dx} \quad (2.64)$$

With the use of this parameter, the momentum-integral equation is nondimensionalized by multiplying it by $U_e \theta / \nu$:

$$\frac{\tau_w \theta}{\mu U_e} = \frac{U_e \theta}{\nu} \frac{d\theta}{dx} + \lambda(2 + H) \quad (2.65)$$

where μ is the dynamic viscosity. The left-hand side of this equation is a friction factor, S , which Thwaites showed to correlate with λ on the basis of published laminar boundary-layer data. Likewise, Thwaites demonstrated correlation of the shape factor, H , with λ . White (2006) provided polynomial alternatives for Thwaites' tabulated forms of $S(\lambda)$ and $H(\lambda)$:

$$S(\lambda) \approx (\lambda + 0.09)^{0.62} \quad (2.66)$$

$$H(\lambda) \approx 2.0 + 4.14z - 83.5z^2 + 854z^3 - 3337z^4 + 4576z^5 \quad (2.67)$$

where

$$z = (0.25 - \lambda) \quad (2.68)$$

Eqns. (2.66) and (2.67) are the two necessary closure equations to solve the momentum-integral relation. Since the closure equations are coupled to θ via λ , it is required to solve the set of equations iteratively at a given streamwise location until convergence to the desired level is reached for θ .

2.5.2.2 Turbulent Closure Method

For turbulent flows, the relationship between the boundary-layer parameters change with different velocity profiles, complicating the closure process (White, 2006). The following method of closure for the turbulent momentum-integral relation is provided by White (2006). The first closure equation is an algebraic relation that is

derived using Coles' law of the wake (1956). The integral parameters given by Coles' law of the wake are arranged as follows:

$$\lambda_f = a(\Pi) \frac{H}{H-1} \quad (2.69)$$

where

$$a(\Pi) = \frac{2 + 3.179\Pi + 1.5\Pi^2}{\kappa(1 + \Pi)} \quad (2.70)$$

and

$$\lambda_f = \sqrt{\frac{2}{C_f}} \quad (2.71)$$

$$\text{Re}_\theta = \frac{1 + \Pi}{\kappa H} \exp(\kappa \lambda_f - \kappa B - 2\Pi) \quad (2.72)$$

where,

$$\text{Re}_\theta = \frac{U_e \theta}{\nu} \quad (2.73)$$

is the Reynolds number based on momentum thickness; λ_f is a measure of the local skin friction; Π is Coles' wake parameter; κ is von Kármán's constant, $\kappa = 0.41$; and B is the turbulence law of the wall intercept constant, $B = 5.0$. If Π is eliminated in the above equations, a curve fit for the relation between C_f , H , and Re_θ is obtained:

$$C_f \approx \frac{0.3e^{-1.33H}}{(\log_{10} \text{Re}_\theta)^{1.74+0.31H}} \quad (2.74)$$

White (2006) asserts that this is the most accurate correlation to use in von Kármán integral-based methods.

The second closure equation is an empirical correlation provided by Das (1987), between Coles' wake parameter, Π , and Clauser's equilibrium parameter, β , for the range $0 \leq \Pi \leq 100$:

$$\beta = -0.4 + 0.76\Pi + 0.42\Pi^2 \quad (2.75)$$

Since the above set of closure equations introduce two new parameters, Π and β , that need to be related to C_f , H , and θ , a third equation is needed. Π is related to C_f and H by Eqns. (2.69) and (2.70). By also using the definition of the Clauser parameter, β , the following is obtained:

$$\beta = -\lambda_f^2 H \frac{\theta}{U_e} \frac{dU_e}{dx} \quad (2.76)$$

Eqn. (2.76) is the final closure equation. The closure equations (Eqns. (2.69), (2.74), (2.75), and (2.76)) are coupled to θ , C_f , and H , so it is required to solve the set of equations iteratively at a given streamwise location, until convergence to the desired level is reached for two of the three parameters, namely θ and C_f , since the third parameter is dependent on the other two.

In this chapter, basic concepts and methods related to wind turbine and propeller aerodynamics have been presented. In the following chapter, relevant aerodynamic analysis methods are discussed and the ones that are suitable to be used in the present algorithm are identified.

Chapter 3: Literature Review

In this chapter, available methods to calculate the aerodynamics of wind turbine and propeller blades are discussed. From these methods, suitable ones for the present algorithm are selected and the reasoning behind the selection is justified. Finally, additional effects that may be considered for the aerodynamic analysis of wind turbines and propellers are discussed.

3.1 Analysis Methods for Turbomachinery Performance and Aerodynamics

3.1.1 Momentum (Actuator Disc) Theory

Momentum theory, developed by Betz (1926), is a well known and simple method for calculating the aerodynamics and the power output of an idealized rotary machine that extracts energy from or imparts energy to a fluid, such as wind turbines and propellers. The mathematical development of the theory can be found in Section 2.2.1.

The momentum theory assumes an ideal turbine with no losses; infinite number of blades, uniform thrust over the rotor disc, no friction drag, and no wake rotation (Manwell et al., 2009). Applying these assumptions causes the results predicted by the momentum theory to be far from reality, so the theory can only be used as a first-order method for turbomachinery design (Leishman, 2006). In addition, it is not generally possible to use the momentum theory for aerodynamic analysis since the induction factor a is not known (Leishman, 2006). However, the theory played an important role in the development of more modern techniques.

3.1.2 Blade-Element Theory (BET)

The blade-element theory (BET), is the basis of the current widely used approach for wind turbine and helicopter rotor aerodynamics (Leishman, 2006). The blade-element approach divides the blade into a finite number of segments (elements) along its span and assumes that each blade element acts as a 2D airfoil to produce aerodynamic forces and moments. Although the BET is an important method in rotor aerodynamics, it is not widely used today because a more advanced method, the blade-element-momentum theory, has been developed.

As cited by Leishman (2006), Tokaty (1971) provides a historical overview of the blade-element and blade-element-momentum theories. The BET approach was first suggested by Drzewiecki (Drzewiecki, 1892; 1920) for the analysis of airplane propellers. Lanchester (1915) has also made contributions to the theory, as suggested by Glauert (1935). As further improvements to the basic blade-element theory, the BET and momentum theories were linked together by Reissner (1910; 1937; 1940), de Bothezat (1919), and Glauert (1935) to accommodate the lift-induced velocity or induced angle of attack distribution over the blade span, thereby creating the blade-element-momentum theory (discussed in the following section). The BET was extended to explicitly include the influence of the vortical wake as well as the other blades using an induced angle of attack component, calculated by means of the Biot-Savart law. These ideas were initially developed by Joukowski as described by Tokaty (1971), Glauert (1922), Bienen & von Kármán (1924), and Lock et al (1925). Prandtl & Betz (1919) and Goldstein (1929) developed a prescribed vortex-wake theory for lifting propellers, which was later

extended by Theodorsen (1948). Glauert (1935) reviews how the early theory applies to propellers.

The final working equations of the BET are presented in Section 2.2.2, with explanations of the primary terms that appear in the BET and the blade-element-momentum theory. The full derivation, which can be found in textbooks on rotor aerodynamics, such as the work of Leishman (2006), is not presented in this manuscript for reasons of brevity.

As Leishman (2006) explains, there are two problems associated with the use of the BET. The first problem is that the BET does not provide a way to calculate the induced inflow velocity v_i (or the induction factor, a). It is possible to assume 2D aerodynamics and proceed with the calculation assuming the induced velocity is zero, but this approach deviates from the actual 3D nature of the problem. The second problem is the need for the 2D aerodynamic coefficients of the airfoil (C_l and C_d) to be known in advance. Since the aerodynamic coefficients usually cannot be expressed as analytical functions, a table look-up approach that uses data from a wide range of angles of attack and Reynolds numbers is needed. Because of these reasons, BET is not exclusively used for the aerodynamic analysis of wind turbine and propeller blades. However, it is the basis of the current and most widely-used approach, the blade-element-momentum theory (BEMT).

3.1.3 Blade-Element-Momentum Theory (BEMT)

The blade-element-momentum theory (BEMT) is a hybrid method that relies on the fundamental relationship between the circulation and momentum theories of lift (Leishman, 2006). Its key difference from the BET is its ability to estimate the induction

factor (or induced velocity) along the blade, using certain assumptions (Leishman, 2006). With the use of the induction factor, aerodynamic loads along the blade span are calculated. According to Leishman (2006), the BEMT has proven to be a very useful method in understanding how some of the key design factors influence wind turbine performance. However, the predictions of the BEMT are generally accurate under conditions where the wind is normal to the plane of rotation, i.e., when the turbine is not yawed (Leishman, 2006). This limitation, which occurs due to the non-axisymmetric and 3D nature of the flow when the turbine is yawed, does not affect the wind turbine aerodynamic calculations for constant wind directions, as wind turbines are not yawed under normal operating conditions. For more accurate aerodynamic calculations under yawed conditions, free-vortex methods may be used (Leishman, 2006), which are briefly explained in Section 3.2.3.1.

As Leishman (2006) explains, the BEMT is based on the combination of a momentum balance on successive annuli of the turbine disc and a blade-element treatment of the sectional aerodynamics. This combination allows solving the inflow velocity of the turbine at each blade element, since it is possible to calculate the lift-induced velocity with the BEMT. The BEMT treats each blade element as a 2D airfoil, which assumes that spanwise flow and spanwise loading gradients are small. According to Leishman (2006), this is a reasonable assumption for unstalled and unyawed conditions, due to good agreement between the predictions of the BEMT and the more advanced free-vortex methods, which incorporate the 3D effects. The phenomenon of spanwise (radial) flow, which occurs under stalled conditions and conflicts with the underlying assumptions of the BEMT, is explained in Section 3.4.3.

A full derivation of the BEMT for a wind turbine configuration can be found in Section 2.2.2. The BEMT provides sufficiently accurate predictions to be used during the design optimization of a wind turbine, and therefore it is the simplest, most general method available today (Leishman, 2006). However, it requires some aerodynamic characteristics (such as the lift slope and the drag coefficient) of the 2D airfoil to be known in advance. It also falls short of calculating the exact aerodynamic loads –lift, drag and pitching moment– imposed on each blade element, as the respective coefficients are not explicitly calculated. These coefficients can be obtained by using the angle of attack of the blade element and 2D airfoil data. It is also not possible to use the BEMT for a new airfoil design, for which the aerodynamic properties are not known. Therefore the BEMT is not a stand-alone method for turbomachinery blade design.

3.1.4 Navier-Stokes Equation Solutions

Contrary to the simpler methods discussed in the preceding sections, there are computational fluid dynamics (CFD) methods that provide more accurate predictions at the cost of time and computation power. These methods generally solve the Navier-Stokes equations for the entire flowfield, therefore treating the flowfield as entirely viscous. Pope (2000) provides a summary of the common methods used to solve the Navier-Stokes equations.

Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) are used to simulate turbulent flow, which is typically the type of flow over a notable portion of the wind turbine/propeller blade. As Pope (2000) explains, In DNS, the Navier-Stokes equations are solved to determine the velocity and pressure field for all turbulence length

and time scales. Therefore, DNS is the most computationally-expensive method. In LES, the Navier-Stokes equations with small spatial and temporal scales filtered are solved to obtain a velocity field which represents only the large-scale turbulent motions. Therefore, LES is computationally less demanding due to the omission of the computation cost needed to solve for small-scale turbulence effects.

As explained by Pope (2000), the Reynolds-Averaged Navier-Stokes (RANS) method involves the solution of the time-averaged Navier-Stokes equations to determine the velocity and pressure field. In RANS methods, the Reynolds stresses are determined using a turbulence model, either via the turbulence viscosity hypothesis, or from modelled Reynolds-stress transport equations (Pope, 2000). In recent years, some RANS models including the works of Sorensen et al. (2002) and Fletcher et al. (2009), have been used to provide significant insight into the aerodynamics of wind turbines.

The methods that are described in this section can be used to solve the full flowfield around the airfoil with higher accuracy compared to the previously-discussed methods. As mentioned previously, these methods treat the entire flowfield as viscous; however, it is generally accepted that such a treatment is not always necessary to obtain predictions that are sufficiently accurate for design optimization (Leishman, 2006). The viscous effects are only dominant in the boundary layer for an attached flow (White, 2006). Therefore, for the region far away from the airfoil surface, using solutions that treat the entire flowfield as viscous are not necessary. However, an aerodynamic analysis of a fluid-immersed body must properly account for viscous effects to calculate the viscous drag, which is caused by the combination of the skin-friction drag and the low-pressure region occurring in the wake of the body (Katz & Plotkin, 1991).

At an early design stage for a wind turbine blade or propeller, the time-intensive methods discussed in this section cannot be used due to the iterative nature of the design process and the need to quickly evaluate the effect of design changes. Nevertheless, provided the computation power is available, these methods can be used to perform a detailed analysis at the final design stage.

3.1.5 Viscous-Inviscid Interaction (VII)

Instead of treating the entire flowfield as inviscid, or having to rely on computationally-expensive solutions which treat the entire flowfield as a viscous flow, an alternative approach is viscous-inviscid interaction (VII). The VII approach treats portions of the flowfield as either inviscid or viscous. The approach uses an inviscid approximation to develop a solution for the flow region assumed to be inviscid, which is coupled to a viscous solution of regions where viscous forces are deemed to be significant. An iterative solution method is usually employed until the solution reaches convergence to a desired accuracy.

Veldman (2009) and Veldman & Coenen (2005) provide a historical overview of the development of VII methods. These methods were facilitated by Prandtl's (1905) discovery of the boundary layer approximation. The boundary layer is influenced by the inviscid pressure distribution over the flow surface. The presence of the boundary layer deflects the inviscid flow streamlines away from the body surface due to mass conservation, and thus makes the body immersed in the fluid "feel" thicker to the inviscid freestream. The resulting effective shape, which accounts for the presence of the boundary layer, is called the displacement body. The displacement body is obtained by

moving the original body surface in the wall-normal direction by a local distance δ^* , called the displacement thickness (defined in Section 2.5.1). The two regions of flow, viscous and inviscid, are shown in Figure 3.1.

Figure 3.1: Subdivision of the flowfield around an airfoil, with an inviscid outer region and a viscous shear layer (exaggerated in thickness).

The oldest VII method is known as the direct method. In this method, the inviscid flow is calculated first, as if there were no boundary layer. The calculation is started with the inviscid flow because the boundary layer (usually) provides only a small perturbation to the inviscid flow. Also, the freestream velocity distribution over the surface, $U_e(s)$, needs to be known for the boundary-layer analysis. The predicted freestream velocity distribution over the surface is then supplied to the boundary-layer analysis to calculate the displacement thickness $\delta^*(s)$, where s represents the streamwise coordinate along the airfoil surface. With the displacement thickness information, the solid-surface geometry is adjusted to account for the presence of the boundary layer, and the inviscid solution is repeated. This process is repeated until convergence to the desired accuracy is achieved. Such methods are generally referred to as *weak viscous-inviscid interaction schemes*, in which a hierarchy between the viscous and inviscid regions is present, and the effects of one region on the other are only felt through the higher-order matching

terms, such as the displacement thickness and the freestream velocity (Rothmayer & Barnett, 1991). A simple flowchart for the direct method is shown in Figure 3.2.

Figure 3.2: Viscous-inviscid interaction scheme for the direct method.

However, when regions of separated flow are present, the boundary-layer calculations break down. This problem arises from a singularity in the boundary-layer equations at the separation point, called the Goldstein singularity. A study of the breakdown process has been presented by Goldstein (1948). At that time, no conclusion on the exact cause of this breakdown was given. Lighthill (1953; 2000) provided some information on the related issue of upstream influence in subsonic and supersonic flows. For supersonic flow, Hayes and Probstein (1959) explained the nature of the interaction near a point of flow separation. They suggested that, in general, it is required to solve the integrated momentum and energy equations for the boundary layer along with the inviscid flow equations for the inviscid surrounding flow simultaneously to overcome the singularity. To overcome the Goldstein singularity for subsonic flow, Catherall and Mangler (1966) presented calculations of a boundary layer with a prescribed displacement thickness.

In the following years, publications by Stewartson (1968), Neiland (1969), and Messiter (1970) introduced the asymptotic triple-deck theory, which provided further information on the Goldstein singularity. Lagre (2009) explains the theory, which is named after an analogy to a ship with three decks. The portion of the boundary layer very close to the wall where the velocity is the smallest is called the “Lower Deck”, and the

rest of the boundary layer is called the “Main Deck”. Perturbations in the “Lower Deck” are transmitted through the “Main Deck”, in which these perturbations act as a displacement of the streamlines. These perturbations in the “Main Deck” may also extend to the upstream flowfield. Then, this displacement of the streamlines is transmitted to the inviscid fluid layer called the “Upper Deck”.

With the developments after the Goldstein singularity was identified, it was seen that the boundary layer has a significant effect on the pressure gradient near separation, instead of the pressure gradient being prescribed through the inviscid flow (Veldman, 2009). Therefore, the hierarchy between the inviscid and viscous flow regions, which allows each of them to be calculated individually (weak viscous-inviscid interaction), breaks down (Lagerstrom, 1975), resulting in what is called *strong viscous-inviscid interaction*. As Rothmayer & Barnett (1991) explain, in strong viscous-inviscid interaction, the effects of the viscous and inviscid regions on one another are not only felt through high-order terms but also through the structures that are present in the flow, thus resulting in the loss of hierarchy.

As Veldman (2009) explains, this loss of hierarchy led to alternative VII methods to be developed in the late 1970s. One method was to use an *inverse method* where the order of information exchange is reversed. A simple flowchart for this method is shown in Figure 3.3a. The method works by inverting both the boundary-layer and the inviscid flow calculations; the inverse boundary-layer analysis uses prescribed displacement thickness data to calculate the freestream velocity required to obtain the given displacement thickness, and the inverse inviscid flow analysis calculates the total thickness of the airfoil when given the freestream velocity calculated by the inverse

boundary-layer analysis. Once the original thickness of the airfoil is subtracted from the total thickness calculated by the inverse-inviscid analysis, the displacement thickness is found. This new displacement thickness is then used to update the prescribed displacement thickness for the boundary-layer analysis and this iterative scheme is continued until convergence to the desired level is achieved. Although this inverse method survives past flow separation, its convergence is very slow (Veldman & Coenen, 2005).

Figure 3.3: Viscous-inviscid interaction schemes for the (a) inverse method and (b) semi-inverse method.

Another method, which was developed by mixing the direct and inverse methods in a *semi-inverse method*, was proposed by LeBalleur (1978) and Carter (1981). In this semi-inverse method, shown in Figure 3.3b, a prescribed displacement thickness is used to solve both inviscid and viscous flow regions, and the difference of the predicted freestream velocity distributions are used to update the displacement thickness via a relaxation-based coupling algorithm. It is called the semi-inverse method because the

boundary-layer calculations are inversed, whereas the inviscid flow is calculated using the direct method.

Yet another method developed by Drela & Giles (1987) and Moses et al. (1978) solves the viscous and inviscid flow equations simultaneously to avoid any hierarchical treatment of both flow regions. The simultaneous solution of the viscous and inviscid flow equations have been used by a number of researchers, including Gilmer & Bristow (1982), and Gilmer et al. (1984), to analyze stalled single- and multi-element airfoils, respectively. Nevertheless, according to Veldman (2009), this approach is complicated and inflexible in the modelling of the flowfield, since it requires both sets of flow equations to be combined into one system. Therefore, attempts were made to approximate this simultaneous approach without complicating the method, and the concept of the interaction law was introduced.

The interaction law suggested by Veldman (1979; 1981) allows the simultaneous solution of both inviscid and viscous flow regions using a sufficiently accurate and simple approximation for the inviscid flow based on the thin airfoil theory (Section 3.2.1). This approximation is used as a boundary condition for the viscous flow analysis, thereby instantaneously accounting for the behaviour of the inviscid flow and reducing the number of iterations required for the viscous-inviscid interactive solution to converge (Veldman, 1981). However, this inviscid flow approximation does not replace the inviscid flow analysis in Veldman's viscous-inviscid interaction scheme. More examples of these interactive calculations have been published by Veldman (Veldman et al., 1990; Veldman, 2001; 2005). However, Veldman himself (2009) states that the

interaction laws used in those works were somewhat complex, because the motivation was to keep the laws' physical contents as close to the real physics as possible.

Viscous-inviscid interaction methods may be considered as the middle ground between pure inviscid analyses (such as BET and BEMT) and computationally-expensive methods such as RANS and DNS, which treat the entire flowfield as viscous. VII methods can provide predictions that are sufficiently accurate to be used for design optimization and they retain the ability to provide a fast solution unlike RANS or DNS. Viscous-inviscid interaction methods are useful in scenarios where a large number of design options must be evaluated while maintaining a degree of accuracy that can only be obtained by accounting for viscous effects. Hence, a VII method has been selected for the purposes of the present study.

In the present study, the direct viscous-inviscid interaction (weak interaction) method is used, since this method is relatively easy to model and adheres to the physics of the flow. The flows studied in the present study have low Mach numbers and thus are assumed to be incompressible, and shockwaves that cause the weak VII approaches to fail are not present. The problem of flow separation is dealt with using empirical data and numerical adjustments that respect the governing physics.

There are two choices to be made when developing a direct viscous-inviscid interaction method; selecting the respective methods to be used for the viscous and inviscid analyses, which are discussed in Sections 3.2 and 3.3, respectively.

3.2 Inviscid Flow Analysis Methods

In this section, an overview of available CFD methods that treat the entire flowfield as inviscid are discussed. Because of the inviscid treatment of the entire flowfield, these methods are not able to capture viscous effects such as skin-friction drag or flow separation. Therefore, they are not capable of modelling the complete aerodynamics of lift-generating bodies by themselves. However, by making the inviscid flow assumption, they allow the Navier-Stokes equations to be greatly simplified and computed faster. The inviscid flow assumption must only be used for the part of the flowfield where viscous effects are not dominant, i.e., further away from any solid surface.

The aim of this review is to select a suitable inviscid flow analysis method to be used in the viscous-inviscid interaction scheme identified in the previous section. The selection will be based on the overall accuracy of the method, ease of computation, and suitability to be used in a viscous-inviscid interaction scheme. The methods that are discussed are the thin airfoil theory (TAT), lifting line blade model, and surface singularity (panel) methods.

Most inviscid flow analysis methods such as thin airfoil theory, lifting line blade model, and panel methods use potential flow singularities such as sources, doublets, and vortices located on or inside the airfoil to model the flowfield. Information on these singularities (sources, sinks, vortices, doublets, vortex sheets, etc.) and their effects on the flowfield around them can be found in any fluid mechanics or aerodynamics textbook; examples include the textbooks of Katz & Plotkin (1991) and White (2011).

3.2.1 Thin Airfoil Theory (TAT)

The thin airfoil theory (TAT) models the 2D airfoil as a camber line, on which a vortex-sheet singularity with an unknown strength, $\gamma(x)$, is placed. Details of this method are provided by Glauert (1983). If the vortex strength distribution $\gamma(x)$ over the camber line is known, the pressure distribution over both sides of the airfoil (and thus the freestream velocities) can be calculated. From the pressure distribution, the lift and pitching moment of the 2D airfoil section is determined, as presented in Chapter 4.

As Leishman (2006) explains, in thin airfoil theory, the vortex sheet strength is usually expressed as a Fourier series. Then, by imposing two boundary conditions, the coefficients of the Fourier series are determined. The first boundary condition is the flow tangency on the camber line (or vortex sheet). The second boundary condition is the “Kutta condition”. As explained by Katz & Plotkin (1991), this condition states that the rear stagnation point on an airfoil should be located at the trailing edge.

Since TAT applies the flow tangency condition on the camber line rather than the surfaces of the airfoil, the predicted velocity field may deviate from the actual values in instances of thick airfoils. This limitation is prominent especially near the leading edge, where the geometry of the airfoil has a significant effect on the flow. The leading edge geometry, along with the undisturbed freestream flow direction, determine the exact location of the leading-edge stagnation point and the subsequent acceleration of the flow starting from the stagnation point. Because TAT does not provide this information, which is necessary for boundary-layer analysis, it is not a suitable method to be used in a viscous-inviscid interaction scheme. Moreover, TAT is not able to distinguish between two airfoils that have the same geometric characteristics except for their thicknesses.

Therefore, the effects of changing the airfoil thickness cannot be determined by this method. This limitation also ties into the previous discussion about the displacement thickness being used to represent the change in the “perceived” thickness of the airfoil by the freestream. Since the airfoil is modelled as a camber line, TAT does not allow the airfoil surface to be displaced outwards to account for the presence of the boundary layer. A method that uses the displacement thickness information obtained from a viscous flow analysis to account for the change in the freestream velocity around the airfoil due to the presence of the boundary layer has been suggested by Veldman (1981); however, the method is only used for an approximate solution with limited accuracy. For these reasons, TAT is deemed unsuitable to be used in the present algorithm.

3.2.2 Lifting-Line Blade Model

Another simple method that can be used to model lift-generating bodies with finite span is the lifting-line blade model. This approach treats the blade as a series of 2D blade elements, similar to the BET or BEMT. The key difference being that the finite-span blade and the near-wake are modelled as line vortices. The blade is replaced by a “lifting-line”, which is a line vortex that represents the lift generated by the blade via the bound circulation. At the blade tips, the vortex line bends and extends far downstream, representing the wake (Figure 3.4a). The vortex line needs to be extended far downstream because the Kelvin-Helmholtz theorem states that a vortex filament cannot start and end in a fluid; it must extend to the fluid boundary or form a closed path. The vortex line that represents the blade is bound to the aerodynamic centre of the airfoil. The strongest vortices are shed from the blade tips, but some vorticity shed along the span of the blade

is also included in the analysis. Prandtl (1918) succeeded in modelling the trailing wake as a whole vortex sheet, instead of individual vortex lines as shown in Figure 3.4b.

Figure 3.4: (a) Generic lifting-line model for a finite wing. (b) Lifting-line blade model by Prandtl (1918). Reproduced from White (2011).

The goal of this approach is to calculate the induced velocity (or angle of attack) at each blade element, so that the effective angle of attack along the blade span can be found. Details of this method have been provided by Katz & Plotkin (1991), Anderson (2011), Leishman (2006), and White (2011). The effective circulation is zero at the blade

tips and maximum at the centre. The lift generated by the blade is directly related to the circulation via the Kutta-Joukowski theorem:

$$dL = \rho V_{\infty} \Gamma dy \quad (3.1)$$

where dL is the lift force of an infinitesimal blade element, ρ is the density of the fluid, V_{∞} is the undisturbed freestream velocity relative to the blade element, Γ is the circulation around the blade element, and dy is the span of the blade element. The lift force is perpendicular to the undisturbed freestream velocity direction. From Figure 3.4a, it can be seen that since the circulation is zero at the blade tips, the generated lift is also zero. The lifting-line model can approximate this effect.

Leishman (2006) states that the coupling between the wake effects on the blade loading and the circulation that is trailed and shed into the wake allows some 3D effects to be modelled using the lifting-line model. However, because small disturbance approximations are used in this method, the lifting-line model cannot fully model strong 3D flow effects, such as the ones found near the tip and root of the blade. Moreover, because the wakes of rotating blades are not planar, care must be taken when using this method. The lack of a solid airfoil geometry and the lack of pressure distribution information on the airfoil surface make the lifting-line blade model unsuitable for a viscous-inviscid interaction scheme.

3.2.3 Surface Singularity (Panel) Methods

Surface singularity or *panel methods*, developed by Hess & Smith (1964), use the assumption of incompressible, irrotational flow, governed by Laplace's Eqn. (2.41), to model the flow around arbitrary lifting or non-lifting bodies using potential flow

singularities (e.g., sources, vortices, doublets, etc.) distributed over the surface of the body. With the use of panel methods, inviscid flow calculations can be carried out for flows around arbitrary 2D or 3D bodies, including aircraft fuselages, wings, and rotor/turbine blades.

As explained by Katz & Plotkin (1991), the calculation is carried out by first discretizing the body using a number of panels. Generally straight-line panels are used for 2D analyses and rectangular panels are used for 3D analyses. On these panels, potential flow singularities are located in a line (2D) or sheet (3D) type of distribution. For lift-generating bodies, vortices must be used as singularities (Hess & Smith, 1967) because of the relationship between the circulation and lift (the Kutta-Joukowski theorem – Eqn. (3.1)); vorticity bound to the lifting surface must be present in order to produce lift. It is also possible to have more than one type of singularity on panels; for example, a combination of sources and doublets or a combination of sources and vortices.

The unknowns in the solution are the strengths of the singularities placed on each panel. These strengths can be of first or higher order (i.e., the singularity strength over one panel can be constant, linearly changing, or higher order). Also, if more than one type of singularity is used, it is possible to assume a uniform strength for one type of singularity, i.e., the strength of that type of singularity is the same for all panels.

Once the discretization is complete, the velocities in the normal and tangential directions for each panel are calculated in terms of the unknown singularity strengths by combining the influences of the freestream flow, singularities located on other panels, and the singularities on the panel itself that is being considered. Then, the surface flow tangency and Kutta conditions are applied, which provides the final set of algebraic

equations to be solved. Once the final set of algebraic equations are solved for the singularity strengths, the velocity field is calculated by superimposing the freestream velocity and the velocities induced by the singularities at target locations (usually at the centre of each body panel, called the *control point* or the *collocation point*). The pressure distribution over the surface is calculated using the Bernoulli equation for steady incompressible flow. The pressure distribution is then integrated over the surface to determine the lift and pitching-moment coefficients for the body.

The advantage of panel methods is their ability to provide fast and accurate predictions for the inviscid freestream velocity distribution around the airfoil (Katz & Plotkin, 1991). The representation of the solid surface geometry also allows panel methods to be used easily in viscous-inviscid interaction schemes because it is possible to displace the surface panels outwards by the displacement thickness of the boundary layer. Instead of displacing the panels physically (changing their locations in the coordinate system), the boundary conditions can be modified by adding a certain “blowing value” to the normal velocity on each panel to account for the displacement thickness (Katz & Plotkin, 1991).

There are many variations of panel methods (2D vs. 3D, different combinations of potential flow singularities, modelling of the wake, etc.) that have been developed and used in a wide variety of applications, such that an extensive summary is beyond the level of detail that can be provided within this manuscript. A brief overview of 3D and 2D methods is provided in the following sections with some examples of their use.

3.2.3.1 3D Panel Methods

In 3D methods, lift-generating bodies are modelled along with their wakes, which allows the 3D effects such as induced velocity and wake roll-up to be captured. However, such detailed analysis comes at a computational cost. 3D panel methods are explained in detail by Hess & Smith (1967), and Katz & Plotkin (1991).

There are three main challenges to overcome when using 3D panel methods. First, the discretization of the body may be challenging because of the difficulty in modelling a curved surface using flat panels. When using flat panels, gaps may be present between adjacent panels, leading to flow leakage. A remedy to this is to use non-planar panels. Another solution is to increase the number of panels in areas with high curvature, but this approach increases the computation time.

Second, it is difficult to apply the Kutta condition in 3D panel methods. When setting the pressure on the two sides of the trailing edge equal, the Kutta condition becomes quadratic since the pressure term contains the square of velocity. This nonlinear boundary condition has prompted researchers to use linear approximations for the Kutta condition, which can lead to a non-physical pressure mismatch at the trailing edge (Hess, 1990).

Third, possibly the most challenging of all, is modelling the wake and the wake roll-up. Getting the proper wake shape (wake roll-up) can be done in three ways, according to Katz & Plotkin (1991). The first way is to use a prescribed wake shape, usually from experimental data; however, this is limited to the extent of available experimental data. The second way is to use a wake relaxation method, where the initial wake is usually specified as a planar wake extending from the trailing edge. In an

iterative manner, the induced velocity by the wing and the wake is calculated at each wake point, and the wake points are moved according to their local induced velocity, until the desired roll-up is reached. The desired roll-up level is set by the programmer and there is a risk of generating a non-physical roll-up if too many iterations are performed. The third and final way, similar to wake relaxation, is to use a time-stepping method, where the time step is directly related to the motion of the wake.

The latter two methods, wake-relaxation and time-stepping, are also referred to as *free-vortex methods* (FVM), because they allow the wake to be deformed freely. Both methods are computationally demanding, due to the repeated application of the Biot-Savart law, which is used to calculate the velocity induced by a vortex or vortex sheet at a given location (Leishman, 2006). FVM can also be used with other inviscid flow analysis methods, such as the lifting-line blade model, to model the wake. FVM are common practice for helicopter rotor aerodynamics and are making their way into other research areas such as wind turbines (Leishman, 2006). With FVM, it is possible to model the wakes of all the blades on a wind turbine, for example. An example for the use of FVM as part of a 3D panel method is the study by Dossing (2007), in which two different winglet designs for horizontal axis wind turbines are examined. As another example, Sezer-Uzol & Uzol (2009) have used a 3D unsteady vortex-panel method with a time-stepping FVM to analyze the effect of wind shear on wind turbine wake structure and performance.

3.2.3.2 2D Panel Methods

Variations and details of 2D panel methods can be found in a great number of aerodynamics textbooks, including the works of Katz & Plotkin (1991), Kueth & Chow (1998), Cebeci et al. (2005), and Anderson (2011).

2D panel methods are commonly used and their main advantage is their simplicity and computational efficiency. Another advantage is the abundance of experimental 2D airfoil data to verify and validate the method, unlike 3D panel methods. Their main disadvantage is not being able to model lifting bodies in 3D, thus not capturing the effects of lift induced velocities/angle of attack, the wake and spanwise flow. Therefore, the predictions obtained from the panel method need to be adjusted using correlations (e.g., root/tip loss) to be able to predict the aerodynamic characteristics of a finite-span lift-generating body.

Because of its advantages, a 2D panel method with source and vortex singularities is used as the inviscid flow analysis method in the viscous-inviscid interaction scheme of the present study. To account for the presence of the boundary layer, the panels are displaced using the displacement thickness information provided by a viscous flow analysis. A review of viscous flow analysis methods is presented in the following sections.

3.3 Viscous Flow Analysis Methods

In viscous-inviscid interaction methods, the viscous flow analysis is carried out for the boundary layer, which is the region of flow close to the airfoil surface where viscous effects are dominant and an inviscid flow assumption is not suitable. The primary

aim for the viscous flow analysis is to determine the skin-friction coefficient and the displacement thickness. The skin friction is used to calculate the skin-friction drag, whereas the displacement thickness is used to account for the flow displacement effect of the boundary layer in the computation of the inviscid flow away from the airfoil surface.

Readers who are not very familiar with the concept of boundary layers are encouraged to read the background information on boundary layers presented in Section 2.5.1. In the following sections, solutions for the boundary-layer equations are discussed.

3.3.1 Similarity Solutions for the Boundary-Layer Equations

Solutions for the boundary-layer equations have been developed in the form of similarity solutions, especially for laminar flows. The common denominator in these methods is to reduce the partial differential equations governing boundary-layer flow into ordinary differential equations using certain simplifications and assumptions. A comprehensive coverage of these methods has been provided by White (2006).

There are two notable similarity solutions for attached laminar boundary layers. First is the flat-plate laminar solution by Blasius (1907) with the later addition of wall suction and blowing by Schlichting and Bussmann (1943) and Hartnett and Eckert (1957). Second is the Falkner-Skan wedge flows, discovered by Falkner and Skan (1931) and later numerically solved by Hartnett and Eckert (1937). An approximate power-law solution for a zero-pressure-gradient turbulent boundary layer has been provided by Prandtl (1921). Since Blasius' and Prandtl's solutions are only applicable for a flat plate, they are far too simple to be used for a flow over an airfoil since these type of flows have

pressure gradients which have significant effects on boundary layers. Also, these methods do not model flow transition; they are only valid for fully laminar or turbulent boundary layers. However, Blasius' and Falkner & Skan's similarity solutions and Prandtl's approximate solution have been used to verify and validate the viscous flow analysis method used in the present study.

There are also similarity solutions developed for free-shear flows. White (2006) explains the solutions that have been developed for three examples of free shear layers: (1) the free shear layer between parallel moving streams (Lock (1951)), (2) laminar and turbulent jets (Schlichting (1933) and Pai (1954) have studied laminar jets; Görtler (1942), George (1989; 1995) and George & Castillo (1997) have studied turbulent jets), and (3) the analysis of laminar and turbulent wakes far downstream of a solid body. Similarity solutions with farfield approximations for plane laminar wakes have been provided by Tollmien (1931) and Berger (1971). However, similarity solutions for immediately downstream locations up to three body lengths are not possible as the wake is still developing (i.e., it does not remain self-similar) (White, 2006). Also, the wakes that are found in modern lifting body applications are almost always turbulent. Similarity solutions for these turbulent wakes have been studied by Sreenivasan and Narasimha (1982) and Wygnanski et al. (1986). Wygnanski et al. (1986) state that in their studies, the turbulent wakes do not become self-similar until hundreds of momentum thicknesses downstream. Because similarity solutions are not readily applicable for the near-wake region, which is important in a detailed analysis of the wake, they pose a discontinuity in the viscous flow analysis. Also, the aim of the present study is to provide an algorithm to calculate the aerodynamic coefficients without having to explicitly model the complex

behaviour of the wake. Therefore, approaches other than similarity solutions are considered in the present study.

3.3.2 Alternative Analysis Methods for Boundary Layers

Because similarity solutions are valid only for certain velocity profiles, they are not usable in most situations due to pressure gradients that make the flow non-self-similar (White, 2006). According to White (2006), there are three approaches that can be used to analyze non-self-similar flows. The first approach is analytic continuation by series expansion, which is now obsolete due to the amount of computational power available to researchers (White, 2006). Two of the notable series expansion methods are the linearly decelerating fluid model developed by Howarth (1938) and a flat plate with uniform wall suction, studied by Iglisch (1944). The second approach is to use integral methods, which use partially-integrated forms of the continuity, momentum, and energy equations to solve ordinary differential equations instead of more complex partial differential equations. The third approach is to use a finite difference or finite element method to numerically model the boundary-layer equations.

Among these three possible approaches, integral methods provide accurate approximate solutions to any 2D steady incompressible boundary layer flow with pressure gradients and are computationally efficient. These methods are discussed in more detail in the following section.

3.3.2.1 Integral Methods

White (2006) explains that partially integrated forms of the boundary-layer equations (continuity, momentum, and energy) are used in integral methods, since their integrated forms are much simpler to solve compared to the full boundary-layer equations, which are partial differential equations. To derive these integral equations, the partial differential boundary-layer equations are integrated across the thickness of the boundary layer, and therefore the velocity in the wall-normal direction and the continuity equation are eliminated in the process. There are two well-known integral equations: the momentum-integral relation developed by von Kármán (1921), and the thermal energy integral relation, studied by Frankl (1934), which relate the momentum loss due to skin-friction drag and the wall heat transfer rate to the streamwise distance, respectively. There is also a less common integral equation, called the mechanical energy integral relation, introduced by Leibenson (1935), which relates the work done by the fluid to the streamwise distance.

The momentum-integral method is used as the viscous flow analysis method in the present algorithm, coupled with the 2D panel method as the inviscid flow analysis method. The full derivation of the momentum-integral relation can be found in Section 2.5.2. There is a need for two closure equations to solve the relation, because there are three unknowns in the equation (assuming the freestream velocity is known *a priori* from the inviscid flow analysis). These closure equations are different for laminar and turbulent flows, since the flow physics are different in these two instances. The closure equations are generally empirical algebraic relationships. The most widely used laminar closure method has been suggested by Thwaites (1949). For turbulent closure,

the first supplementary relation comes from Coles' (1956) wall-wake law, with correlations provided by a number of researchers including Ludwig and Tillmann (1950), Felsch et al. [in Kline et al. (1981)] and White (2006). The second empirical closure relation has been provided by Das (1987). Another alternative for the second relation is to use a differential equation instead of an algebraic relation, called the entrainment relation. This was suggested by Head (1958) and also studied by Ferziger et al. (1982). Using a differential equation for closure further complicates the solution. Moreover, the agreement of the entrainment method with experimental results for favourable pressure gradients is not good (White, 2006). Therefore, empirical algebraic relations for turbulent closure are preferred over the entrainment relation in the present study. The details of the selected closure methods for laminar and turbulent boundary layers are explained in Sections 2.5.2.1 and 2.5.2.2, respectively.

There are two shortcomings of the momentum-integral equation when used for transitioning flows over lifting bodies. First is the lack of transition detection and the momentum-integral relation's inability to calculate transitioning flows, since it can only be used to calculate fully-laminar or fully-turbulent flows. The second weakness is the singularity at the flow separation location, regardless of the boundary-layer flow being laminar or turbulent. The solution cannot be continued past the separation point. These two problems have prevented researchers to agree on a widely-accepted and general scheme in which the momentum-integral equation is used. The present study strives to overcome these difficulties by using a state-of-the-art transition model and experimental results that are applicable to the types of steady incompressible flow and the Reynolds

number range of the present study, while maintaining a simple and computationally-efficient solution.

3.3.3 Flow Transition

The fluid-flow phenomenon in which the flow regime changes from a laminar to a turbulent state is called flow transition. A detailed review of the mechanics of flow transition is beyond the scope of this study, however information on shear-layer instability and transition can be found in the textbooks of Schlichting and Gersten (2000), Chandrasekhar (1961), Schmid and Henningson (2001), and in the report of Mack (1984).

The main purpose for analyzing flow transition is to determine the transition inception and completion locations, so that the correct closure method of the momentum-integral relation (laminar or turbulent) is used in the corresponding flow regions. Use of the correct closure method is important because the behaviour of laminar and turbulent flows are quite different from one another; for example, turbulent flows generally generate more skin friction than laminar flows (White, 2006). Another example is the growth rate of the displacement thickness being different between laminar and turbulent flows, which affects the interaction between the viscous and inviscid regions of flow.

3.3.3.1 Modes of Flow Transition

As Roberts (2005) explains, researchers have classified boundary-layer transition into three modes; natural, bypass, and separation-bubble transition. The following is a summary of the discussion provided by Roberts & Yaras (2005b), and Roberts (2005) on

the three transition modes. The primary mode for attached flows, where the freestream turbulence is low or moderate and the surface is smooth, is natural transition. After a certain critical Reynolds number based on momentum thickness, $Re_\theta = U_e \theta / \nu$, is reached, some disturbances in the boundary layer are amplified, forming two-dimensional instability waves that are named after Tollmien and Schlichting (Schlichting & Gersten, 2000; Tollmien, 1929). These Tollmien-Schlichting (T-S) waves experience a linear growth stage, after which the disturbances start to increase exponentially in a three-dimensional manner, gradually creating a turbulent flow regime.

When increased levels of freestream turbulence, high surface roughness, or other sources of large disturbances are present, the large amplitude disturbances may lead to the linear growth stage of disturbances being bypassed, resulting in what is called bypass transition (Morkovin, 1969). In such cases, there is no two-dimensional growth of T-S waves; instead, three-dimensional turbulence appears almost immediately in the boundary layer. Due to the disturbances being large, bypass transition usually starts further upstream than natural transition. Bypass transition is a common occurrence in turbomachinery applications, where the freestream turbulence intensity is generally quite high (Roberts, 2005). Any build-up at the leading edge of wind turbine and propeller blades would also create additional surface roughness, promoting bypass transition.

Separation-bubble transition occurs in flows, where as the name suggests, transition takes place after laminar flow separation. If the laminar boundary layer is subjected to an adverse pressure gradient or an obstacle before the onset of transition, the shear layer may separate. The highly inflectional velocity profile of the separated shear layer is inviscid unstable, thus triggering transition onset shortly after separation. The

enhanced mixing in the transitioning separated shear layer promotes reattachment, thus often yielding a separation bubble.

Models to predict flow transition have been developed over the years. Expectations from an ideal transition model would be to accurately predict the transition onset and completion locations, by taking the factors that influence flow transition into consideration, which are discussed in the following section.

3.3.3.2 Factors Influencing Flow Transition

A number of factors affect the transition behaviour in a shear layer. They are, as described by Roberts and Yaras (2005b), the Reynolds number, freestream turbulence intensity (Blair (1982); Volino & Hultgren (2001)), turbulence length scale (Johnson & Ercan (1997); Roberts & Yaras (2003)), streamwise pressure gradients (Fraser et al. (1994); Gostelow & Blunden (1989); and Sharma et al. (1982)), streamline curvature (Kim et al. (1992); Schultz & Volino (2003)), periodic impingement of wakes (Stieger & Hodson (2004); Dong & Cumpsty (1990); and Schobeiri & Radke (1994)), and surface roughness (Cummings & Bragg (1996), Pinson & Wang (1997), and Roberts & Yaras (2005a)). These factors affect the transition mode, the transition inception and completion locations, and the fraction of time the flow is turbulent in the flow during transition.

The influence of freestream turbulence intensity, pressure gradient, and surface roughness were discussed in the preceding section. The details of how each factor affects flow transition is beyond the scope of this study; the reader may refer to the works cited above for more information. In the following section, existing transition models are

reviewed to identify one that is suitable for the present algorithm, which ideally accounts for all of the primary factors that affect flow transition.

3.3.3.3 Transition Models

Numerical solutions such as DNS, LES, and RANS utilizing low Reynolds number turbulence models can be used to model flow transition. However, the use of such methods is beyond the level of computational complexity aimed in the present study. There are also semi-empirical algebraic transition models that have been developed and improved over the years. As the name suggests, these methods rely on mathematical correlations calibrated through experimental data.

Emmons (1951) proposed that transition in attached boundary layers is the result of the production of *turbulent spots*, which are small patches of turbulent flow that spread in the streamwise, spanwise, and cross-stream directions as they are convected downstream, where they merge with each other to form a fully-turbulent boundary layer. Emmons showed that the inception, convection, and spreading rates of turbulent spots determine the length of the transition zone. Experimental studies by Fraser et al. (1994) and Gostelow et al. (1994), among others, show that the rate of turbulent spot production is affected by the freestream turbulence intensity and streamwise pressure gradient. In addition, the work of Pinson & Wang (1997) indirectly shows the effect of surface roughness on turbulent spot production. Other studies that investigate the relationship between flow transition and turbulent spots include the works of Schubauer & Klebanoff (1955), Narasimha (1957), Johnson & Fashifar (1994), Gostelow et al. (1996), Zhong (2002), and Chong & Zhong (2005).

Emmons (1951) also proposed a key parameter, *intermittency* (γ), which is the probability of a point in the flowfield being contained by a turbulent spot at any given time. Thus, in the laminar region of the boundary layer, $\gamma = 0$ and in the fully turbulent region $\gamma = 1$, except for locations in the boundary layer where turbulence is always intermittent, that is, the freestream edge of the boundary layer. A global intermittency model was proposed by Narasimha (1957), with following adjustments by Chen & Tyson (1971), Savill (1993a; 1993b), Solomon et al. (1995), Gostelow et al. (1996), D'Ovidio et al. (2001), and Roberts & Yaras (2005b). Johnson & Ercan (1997) have also suggested an alternative intermittency model which focuses on elevated freestream turbulence conditions ($Tu > 2\%$).

The first studies that investigate transition inception and completion in attached flows by Hall & Hislop (1938) and Schubauer & Skramstad (1948), showed that the freestream turbulence intensity has a significant effect on the boundary layer over a flat plate. Later, Abu-Ghannam & Shaw (1980) and Martin et al. (1978) expanded the range of freestream turbulence levels that were fairly low in the previous studies, and included the effects of the streamwise pressure gradient on the boundary layer. Mayle (1991) and Hourmouziadis (1989) concluded that for freestream turbulence intensities over about 3% the streamwise pressure distribution has minimal effect on the transition inception location compared to the turbulence intensity.

For transition in short separation bubbles (separation bubbles that have only a local effect on the pressure distribution and consist of only one recirculation zone (Gaster, 1969)), studies including Mayle (1991), Roberts (1975), Davis et al. (1987), Hatman & Wang (1999), Yaras (2002), and more recently Roberts & Yaras (2005b) have

provided relationships between the location of transition inception and the state of the boundary layer at the separation location. Predicting the transition length in separation bubbles is more difficult because transition occurs very rapidly (Roberts, 2005). Initially, such as in the studies by Horton (1969) and Roberts (1980), point transition was assumed. This was a reasonable assumption, except in turbomachinery (particularly low pressure turbines), where the length of separation bubbles may be significant, up to 30% of the suction surface of a turbine blade (Roberts, 2005). Correlations have been suggested to predict the reattachment location of the flow, since the reattachment location is closely related to the completion of transition, and also the reattachment location is a better parameter to use in the analysis of the turbulent boundary layer downstream of the reattachment location (Roberts, 2005). These correlations have been provided by Hatman & Wang (1999), Volino & Hultgren (2001), and Yaras (2002). Another method to calculate the transition length in separation bubbles is based on calculating the intermittency distribution in the transition region assuming the flow reattachment occurs where transition is complete ($\gamma = 1$). Such intermittency correlations have been provided by Volino (2002), Gostelow & Thomas (2005), and Roberts & Yaras (2005b).

In the present algorithm, the transition model provided by Roberts & Yaras (2005b) is used to predict the transition inception and completion locations (or in the case of separation bubbles, the reattachment location) and the intermittency distribution in the transition region. This model provides a unified treatment for both attached and separated-flow transition in all three modes, including the effects of streamwise pressure gradient and freestream turbulence intensity, while also accounting for the effect of

surface roughness on separation-bubble transition. The details of this model and how it is implemented with the chosen viscous flow analysis method is presented in Chapter 4.

3.3.4 Flow Separation

As stated in Section 3.3.3.1, if the boundary layer separates in a laminar state, transition will very likely develop in the separated shear layer, usually resulting in the reattachment of the shear layer. Turbulent boundary layers are known to be generally more resistive to separation compared to laminar boundary layers due to the turbulence activity energizing the boundary layer with the momentum in the freestream (White, 2006). The effects of flow separation may be small if it occurs near the trailing edge (usually at small angles of attack), or quite large, causing stall at high angles of attack due to separation occurring at a more upstream location, resulting in increased pressure drag and a loss of lift force (Katz & Plotkin, 1991).

Separated flows are inherently one of the most difficult types of flow to model due to their unsteady nature, and the analytical methods for determining the resulting pressure distribution due to separated flows have not reached a state in which these methods can confidently be used for engineering purposes (Katz & Plotkin, 1991). Moreover, there is a substantial lack of experimental data on boundary-layer parameters such as the skin-friction coefficient and displacement thickness past the turbulent separation point.

Katz & Plotkin (1991) indicate that some time-dependent inviscid methods that model the separated shear layer as a sheet of discrete vortices exist, which have been summarized in the works of Leonard (1980) and Sarpkaya (1989). However, a time-

dependent analysis is beyond the scope of the present study. A simpler and commonly used approach to modelling regions of separated flow is to assume a uniform pressure distribution between the separation point and the airfoil trailing edge, which is observed in the time-averaged pressure distributions on numerous stalled airfoils (Katz & Plotkin, 1991). The application of this approach to the present algorithm is discussed in Section 5.3.1.2.

Apart from the alternative viscous-inviscid interaction methods (discussed in Section 3.1.5, such as the inverse method and semi-inverse method), which may be used to overcome the singularity in the boundary-layer equations at the separation point, some researchers have proposed inner-variable integral boundary-layer equations that can be used past the separation point. One such study is the work of Das & White (1986), where the authors present a single differential equation, which can be solved past the separation point, and can predict the negative skin-friction coefficient values in the recirculation zone for a number of flows. However, the predictions for the displacement and momentum thicknesses past the separation point are not satisfactory (Das & White, 1986). Moreover, the complexity of the method's closure equations have been found to cause numerical problems when the method was used in the present algorithm.

A new approach has been used in the present study for predicting the behaviour of the shear layer after boundary-layer separation, since the available methods for modelling separated flows are deemed unsuitable for the weak viscous-inviscid interaction scheme of the present study. This new approach, presented in Section 4.2.2.7, uses experimental data from each airfoil to provide a correlation to predict the displacement thickness growth after separation, as a function of the airfoil angle of attack.

3.4 Additional Aerodynamic Effects for Consideration

3.4.1 Lift-Induced Velocity

The lift induced velocity is an important 3D effect on finite-span lift-generating bodies, which must be taken into account to accurately predict the aerodynamic loads along the span of a blade. The cause of this induced velocity is the shedding of the vorticity bound to the lifting body (Katz & Plotkin, 1991). The shed vorticity modifies the freestream approaching the airfoil, changing its velocity magnitude and direction and therefore the effective angle of attack. The added velocity component is called the induced velocity, and the resultant modification in the angle of attack is called the induced angle of attack. The lifting-line method and the blade-element-momentum theory, as described in Sections 3.2.2 and 3.1.3, respectively, are two methods that include the calculation of this induced effect.

3.4.2 Lift Loss at the Blade Root and Tip

The induced 3D effects on a finite blade cause a greater loss of lift force at the blade root and tip because of the close proximity to the trailing vortices that are shed from both ends of the lift generating section of the blade (Leishman, 2006). Prandtl & Betz (1919) provided a simple correction method for calculating the tip- and root-loss effects. The method can be used to approximate the root and tip losses for a finite number of blades, various blade planforms, and twist angles. This correction method is satisfactory for helicopter rotors and wind turbines (Leishman, 2006), and it is also sufficiently accurate for propellers (Glauert, 1935; Wilson et al., 1976). This method is

used to calculate the 3D wing aerodynamic loading using the 2D aerodynamic data obtained from the viscous-inviscid interaction method described in Chapter 4.

3.4.3 Unsteady Aerodynamic Effects

According to Leishman (2002), unsteady aerodynamic effects pose a great challenge to researchers due to their inherent difficulty to be modelled, and researchers including Robinson et al. (1995), Huyer et al. (1996), Schreck et al. (2000), and Leishman (2002) have established that wind turbines operate in relatively unsteady flow conditions most of the time. These unsteady effects are generally manifested as a result of blade flapping and twisting, and rapid and frequent changes in the freestream conditions. The main aerodynamic sources of unsteady behaviour on wind turbines are temporal variations in wind speed, wind shear, yaw, tower shadow, wind turbulence, wake dynamics, and blade/wake interactions (Leishman, 2002).

To this date, the scientific community has not come to a complete understanding of unsteady airfoil behaviour, even in attached flow conditions which are somewhat easier to analyse (Leishman, 2006). For example, in the report by Simms et al. (2001), a blind comparison of 19 different wind turbine modelling methods were made against the results from the Unsteady Aerodynamics Experiment (UAE) Phase VI (Hand et al., 2001) conducted by the National Renewable Energy Laboratory (NREL) in the NASA-Ames wind tunnel using a full scale model of a small wind turbine, under unyawed operating conditions. The wide range of methods that were tested include variations of the blade-element-momentum theory (BEMT), various single and multi-body aeroelastic methods, Navier-Stokes methods, and prescribed and free-vortex methods. The turbine power

prediction by these methods ranged from 25% to 275% of the measured power, and the blade bending-force predictions ranged from 60% to 150% of the measured force. This result shows that the current level of aerodynamic analysis techniques needs to be improved greatly to fully understand the unsteady aerodynamic behaviour of wind turbines.

Unsteady aerodynamic effects can cause dynamic stall, which refers to the phenomenon where flow separation and stall on a rotor blade occurs in a more dynamic or time-dependent manner (Leishman, 2006). It is important to note that wind turbine blades are also subjected to unsteady airloads in conditions where dynamic stall is not present, and even in fully-attached flow conditions (Leishman, 2002). According to Leishman (2006), there are two flow effects that cause the onset of flow separation to be delayed resulting in dynamic stall. The first is the reduction in the streamwise adverse pressure gradients compared to steady conditions at the same angle of attack, due to the unsteadiness of the flow resulting from the vorticity that is shed into the wake from the trailing edge of the airfoil. The second is the coupled effect of centrifugal and Coriolis forces on the boundary layer on a rotating blade (i.e., a radial (or spanwise) flow effect).

Researchers including Himmelskamp (1950), Young & Williams (1972), Madsen & Christensen (1990), Snel (1991), Narramore & Vermeland (1992), Dwyer & McCroskey (1971), Robinson et al. (1999), and Schreck et al. (2000; 2001) have studied the 3D effects on rotating blades operating near stall. These unsteady effects cause the power output of rotating blades under stalled conditions to be higher than the values calculated using quasi-steady analysis methods. When compared to helicopter blades, the effects of radial flow on wind turbine blades are much more significant due to their

higher lift coefficients (Leishman, 2006). The experiments of Dwyer & McCroskey (1971) on helicopter rotors suggest the onset of flow separation is delayed to a higher angle of attack due to favourable spanwise effects on the development of the boundary layer, thus increasing the thrust output of the rotor. As a result of this behaviour, approximate methods have been developed to model the observed 3D “*stall delay*” effects by numerous researchers, including Corrigan & Schillings (1994), Du & Selig (1998), Snel et al. (1994), and Lindenburg (2004). Leishman (2006) states that a highly accurate approach for modelling unsteady effects on wind turbines is still missing. Therefore to model unsteady effects, one must either use the approximate stall delay models, or resort to advanced CFD methods, which require considerably higher computation time and power.

In the present algorithm, the Corrigan-Schillings (1994) and the Lindenburg (2004) stall delay models, described in Sections 3.4.3.1 and 3.4.3.2, respectively, are used. The Corrigan-Schillings model is selected due to its extensive usage, and the Lindenburg model is selected because it was developed using the UAE – Phase VI data, which provided unsteady flow conditions in a controlled wind tunnel environment.

3.4.3.1 Corrigan-Schillings Stall Delay Model

As described by Tangler and Selig (1997), the Corrigan-Schillings (1994) model defines the location of the separation point on the blade in terms of a shape factor, c/y , which is the local solidity of the blade, where c is the chord length and y is the radial distance from the rotation axis. The model uses the following empirical relation between

an assumed negative-linear velocity gradient in the streamwise direction on the suction surface of the airfoil, K , and the location of the separation point, c/y :

$$\frac{c}{y} = \frac{0.1517}{K^{1.084}} \quad (3.2)$$

The following relationship for the stall delay angle of attack, $\Delta\alpha$, is derived partially from analytical considerations based on the study by Banks & Gadd (1963), and mostly from helicopter rotor and propeller test data:

$$\Delta\alpha = (\alpha_{C_{l_{max}}} - \alpha_0) \left(\left(\frac{K}{0.136} \frac{c}{y} \right)^n - 1 \right) \quad (3.3)$$

where $\alpha_{C_{l_{max}}}$ is the stall angle and α_0 is the zero-lift angle of attack. According to Tangler & Selig (1997), Eqn. (3.3) implies that airfoils with a greater stall angle have a larger radial pressure gradient that results in an increased stall delay, compared to airfoils with a lower stall angle. According to Corrigan & Schillings (1994), the exponent n can be set to values between 0.8 and 1.6 to get a good agreement with most test data, and $n=1$ gives a good estimation for many cases. Therefore $n=1$ is used in the present algorithm. The stall delay angle of attack, $\Delta\alpha$, is used to modify the lift coefficient, C_l , calculated by the 2D viscous-inviscid interaction method of the present study using:

$$C_{l,rot} = C_l + C_{l_\alpha} \Delta\alpha \quad (3.4)$$

where $C_{l,rot}$ is the lift coefficient under rotating conditions and C_{l_α} is $dC_l/d\alpha$ for the airfoil.

3.4.3.2 “Centrifugal Pumping” Model of Lindenburg

The term “centrifugal pumping” in Lindenburg’s (2004) model to account for the change in aerodynamic coefficients under rotating conditions refers to the mechanism by which the air in the separated-flow region flows to outer radial locations due to the spanwise pressure gradient and the chordwise negative pressure gradient on the airfoil. Lindenburg argues that most of the earlier stall delay models do not account for the increase in the drag coefficient under rotating conditions, as observed by Himmelskamp (1947), Milborrow (1985), Viterna & Corrigan (1982), and in the NREL Unsteady Aerodynamics Experiment (UAE) – Phase VI (Hand et al., 2001).

The model is based on calculating the additional normal force on the suction side of the airfoil due to the Coriolis acceleration acting on the separated-flow region at the trailing edge, which is present in a rotating coordinate system and has a centrifugal-loading effect on separated volumes of flow. The tangential induced velocity due to rotation is assumed to be small, and the Coriolis force created by the Coriolis acceleration is expressed in terms of the tangential velocity (Ωy) of the blade, where Ω is the angular velocity of the blade and y is the radial distance from the rotation axis. The Coriolis force is then nondimensionalized using the dynamic pressure of the undisturbed freestream approaching the blade element and merged into the definition of the inflow angle, ϕ , shown in Figure 2.3. The final equation is formulated to calculate the increase of the force coefficient in the chord-normal direction, C_n , which in turn results in an increase in both the lift and drag coefficients:

$$C_{n, rot} = C_n + 1.6 \left(\frac{c}{y} \right) (1 - f_{sep}) \cos^2 \phi \quad (3.5)$$

where $C_{n, rot}$ is the normal force coefficient under rotating conditions, f_{sep} is the nondimensional chordwise location of flow separation, and ϕ is the inflow angle of the flow approaching the blade element. The ‘scaling factor’ of 1.6 in Eqn. (3.5) is similar to the calibration factors present in other stall delay correlations and is airfoil dependent, and comes from comparisons with the UAE-Phase VI rotor. Lindenburg indicates that more research is necessary to determine a scaling factor that is suitable to use for most rotor calculations.

3.4.4 Rotor Wake Effects

In the design of wind farms, the effect of the rotor wake on another turbine located downwind must be considered. According to Hau (2006), operating in the wake of another turbine has significant consequences such as:

- reduced energy output due to the lower mean-flow velocity (velocity deficit) in the wake;
- increased turbulence levels in the wake also increasing the turbulence loading on downwind turbines, which may affect the fatigue strength of the blades despite the lower steady-state loads due to the velocity deficiency in the wake; and
- the rotor wake adversely affecting the blade pitch angle control on downwind turbines.

As Hau (2006) explains, the highest levels of velocity deficiency occur in the near wake region (1~2 rotor diameters downstream) close to the rotor hub, decreasing toward the blade tips. At further downstream locations (3~5 rotor diameters), significant

turbulence is generated at the boundary of the expanding rotor wake, which also mixes with the turbulence and higher wind speed outside the wake, promoting the dissipation of the vortices generated by the rotor. The velocity profile in the far wake region (>5 rotor diameters) has a Gaussian distribution, and the reduction in the velocity deficit in this region is significantly influenced by the turbulence intensity of the surrounding air. Hau (2006) also indicates that the thrust coefficient of the rotor has a significant effect on the velocity deficiency behind the rotor and the extent of the wake.

Vermeer et al. (2003) indicate that larger thrust coefficients increase the turbulence generated in the wake, which decreases as the wake travels downstream. They also indicate that turbulence effects are more persistent than the velocity deficit, such that Hojstrup (1999) and Högström et al. (1988) have found that the velocity deficit at 10~12 rotor diameters downstream is negligible, whereas turbulence effects are still present. It should also be noted that even though the turbulence in the atmospheric boundary layer is non-isotropic, the turbulence in the wake is slightly more isotropic compared to the outside flow (Vermeer et al., 2003).

Wind tunnel experiments have been conducted by researchers including Chamorro & Porté-Agel (2009) and Maeda et al. (2011) to measure the time-averaged velocity profile and turbulence intensity across the wake at various locations. The change in the integral length scale of turbulence in the wake remains to be investigated. The subject of optimum downwind spacing of turbines in a wind farm may be revisited to include the effects of the integral length scale of the turbulence in the wake along with the velocity deficiency and turbulence intensity. Since the integral length scale of turbulence increases with downstream distance, increasing the downwind spacing may

increase the maintenance costs and decrease the turbine life span due to the large-scale turbulence loading on the blades causing fatigue. Decreasing the downwind spacing will likely result in a lower power output; however, a smaller integral length scale of turbulence may result in reduced turbulence loading on the blades.

Many mathematical wake models, which are beyond the scope of the present study, have been developed by a number of researchers, with some of the earliest ones being suggested by Lissaman (1979), Ainslie (1987), and Crespo et al. (1990). Vermeer et al. (2003) provided an extensive survey on wake aerodynamics and wake models.

In the present study, it is assumed that any velocity deficiency in the wind profile approaching the turbine is negligible, since the current practice for the downwind spacing of turbines in wind farms is about 8~10 rotor diameters (Manwell et al., 2009). However, the effects of freestream turbulence intensity and turbulence integral length scale on boundary-layer transition are considered in the present viscous flow analysis (Section 4.2.2).

3.4.5 Wind Shear

The freestream approaching the turbine may have a significant velocity gradient depending on the upstream terrain, since the turbine operates in the atmospheric boundary layer (Leishman, 2006). This velocity gradient may reduce the mass flow through the turbine, reducing its power output, and also cause periodic fluctuations in the aerodynamic loads as the blade rotates through it.

Two well-established models that represent the variation of the time-averaged wind speed with height above ground are the power law and the logarithmic law. The power law is expressed as:

$$V_{\infty}(h) = V_{\infty}(h_{ref}) \left(\frac{h}{h_{ref}} \right)^m \quad (3.6)$$

where h is the height above ground, h_{ref} is a reference height (usually 10 m above ground), and the values of exponent m depending on the type of terrain upstream of the turbine are presented in Table 3.1, as provided by Walker & Jenkins (1997). The logarithmic law includes a roughness length, k , which represents the type of terrain upstream of the turbine:

$$V_{\infty}(h) = V_{\infty}(h_{ref}) \left(\frac{\ln(h/k)}{\ln(h_{ref}/k)} \right) \quad (3.7)$$

Standard values of k are also presented in Table 3.1.

Table 3.1: Atmospheric boundary-layer model coefficients (data from Walker & Jenkins (1997)).

Type of terrain	$k(m)$	m
Open country	0.02	0.12
Rural with few trees	0.05	0.16
Rural with trees and towns	0.3	0.928
Open water	0.001	0.01

The wind profile must be considered when choosing the site, determining the tower height, and sizing the rotor for wind turbines (Leishman, 2006). The effect of wind shear is assumed to be negligible in the present study, since it is considered that most

wind turbines are placed high enough to avoid the most severe velocity gradients near the ground (below 10~20 m). When operating in moderate wind gradients, the greatest difference in mean wind speed over the course of a 360° rotation is experienced by the blade tips, however the resultant difference in the total velocity was calculated to be negligible ($\leq 3\%$) due to the greater rotational speeds at the blade tips being the main contributor to the total velocity.

Leishman (2006) states that the mathematical models of wind characteristics must be improved and integrated into turbine design methods to prevent the uncertainties in wind-speed estimation, which cause premature failure due to the structural loads being under-predicted. He also indicates that the turbulence component of the wind in the axial, lateral, and vertical directions must also be taken into account as they are a source of unsteady loading.

3.4.6 Streamwise Surface Curvature

For the viscous flow analysis, it is also important to know what kind of effect, if any, the streamwise convex curvature of the airfoil has on the development of the boundary layer. A number of studies have been conducted to analyze the effects of streamwise curvature. The work of Liepmann (1943) concludes that for boundary-layer thickness to curvature radius (δ/r) ratios of up to 0.001, boundary-layer transition is not affected by convex curvature. Moreover, Gowda & Narayana (1980) state that in instances where (δ/r) is smaller than 0.02, adverse pressure gradients have a much greater effect on turbulent flow separation than surface curvature. Finally, Umur & Ozaip (2006) show that convex curvatures have a much less pronounced effect on the boundary layer than

concave curvatures. Therefore, one may assume that the convex streamwise surface curvature of typical airfoils, where the δ/r values are less than 0.001, do not have a significant effect on the boundary-layer development.

In this chapter, a weak viscous-inviscid interaction method and the blade-element-momentum theory have been identified as the main methods to be used in the present aerodynamic analysis algorithm and important aerodynamic effects that need to be taken into account have been discussed. In the next chapter, the algorithm for the aerodynamic analysis of rotating blades is described in detail.

Chapter 4: Algorithm for the Aerodynamic Analysis of Rotating Blades

4.1 Introduction

In this chapter, the numerical methods constituting the algorithm developed in the present study for calculating the aerodynamic loading and performance of wind turbine and propeller blades are presented. The algorithm consists of two parts. The first part, presented in Section 4.2, predicts the 2D aerodynamic characteristics of an airfoil for given Reynolds numbers and freestream pressure distribution and turbulence levels using a weak viscous-inviscid interaction method. The second part, presented in Section 4.3, uses the 2D aerodynamic characteristics and utilizes the blade-element-momentum theory with root and tip losses and additional stall delay corrections to calculate the 3D aerodynamics of a rotating blade. Calculation of the rotor performance parameters is presented in Section 4.4. The flow around the blade is assumed to be steady in time, incompressible, and within a range of typical Reynolds numbers based on chord length for wind turbines and small-scale UAV propellers ($Re_c = 1 \times 10^5 \sim 5 \times 10^6$). Appendix A contains a flowchart of the complete algorithm that includes the numerical methods presented in this chapter.

4.2 A Weak Viscous-Inviscid Interaction Method for Steady, 2D, Incompressible Flow around an Airfoil

The method described in this section uses a direct (weak) viscous-inviscid interaction (VII) approach, as explained in Section 3.1.5. The inviscid flow analysis, which uses a 2D source-vortex panel method is described in Section 4.2.1. The viscous flow analysis, which uses the momentum-integral method, is described in Section 4.2.2.

The calculation of the 2D aerodynamic coefficients is presented in Section 4.2.3, and the interfacing of the inviscid and viscous solutions, along with a summary of the full 2D aerodynamic calculation procedure, is presented in Section 4.2.4.

4.2.1 Inviscid Flow Analysis Using a 2D Source-Vortex Panel Method

The inviscid flow analysis focuses on the region away from the airfoil surface where viscous effects can be neglected to calculate the velocity field around the airfoil. For this, a 2D Hess-Smith-type panel method (1964), described by Cebeci et al. (2005), is used.

4.2.1.1 Discretization for the 2D Panel Method

An airfoil is considered with a relative undisturbed freestream velocity of V_{∞} , at an angle α with the chord line, as shown in Figure 4.1. The 2D coordinate system, $x'-y'$, is attached to the airfoil chord line. The upper and lower surfaces of the airfoil, which can either be given as analytical functions or, more commonly, as a set of coordinates, are described as $Y_u(x')$ and $Y_l(x')$, respectively. The distance of any point (x', y') in the flowfield to an arbitrary point b on the airfoil surface is r . The unit vectors normal and tangent to the surface at any point b are $\bar{\mathbf{n}}$ and $\bar{\mathbf{s}}$, respectively. The angle between the x' axis and $\bar{\mathbf{s}}$ is θ .

$$\bar{\mathbf{n}} = -\sin \theta \bar{\mathbf{i}} + \cos \theta \bar{\mathbf{j}} \quad (4.1)$$

$$\bar{\mathbf{s}} = \cos \theta \bar{\mathbf{i}} + \sin \theta \bar{\mathbf{j}} \quad (4.2)$$

with $\bar{\mathbf{i}}$ and $\bar{\mathbf{j}}$ as the unit vectors in the x' and y' directions, respectively.

Figure 4.1: Terminology for a 2D panel method. Reproduced from Cebeci et al. (2005).

The airfoil surface is represented by a finite number of flat panel elements. There are N_{pan} number of panels, defined by $N_{pan} + 1$ boundary points. In panel methods, it is customary to number the boundary points starting from the lower surface at the trailing edge, and proceed clockwise to end at the trailing edge on the upper surface. It should be noted that the points (x'_1, y'_1) and $(x'_{N_{pan}+1}, y'_{N_{pan}+1})$ are identical and represent the sharp trailing edge. The panel element between boundary points (x'_i, y'_i) and (x'_{i+1}, y'_{i+1}) is called the i -th panel, and the midpoints of the panels are called the *control points* or *collocation points* (x'_m, y'_m) . All unit normal vectors, $\bar{n}_i$, are directed from the airfoil surface into the fluid and all unit tangential vectors, $\bar{s}_i$, are directed from the i -th boundary point to the $(i+1)$ -th boundary point, making the angle θ with the x' axis. Since θ is measured from the positive x' axis to the unit tangential vector $\bar{s}_i$, θ is negative for some of the panels. When establishing the direction of $\bar{n}$ and $\bar{s}$ at a boundary point i , the angles of the two adjacent panels, θ_i and θ_{i-1} , are averaged.

The velocity vector at any point (x', y') in the flowfield is represented by $\vec{V}$, with

$$\vec{V} = \vec{V}_{\infty} + \vec{v}_i \quad (4.3)$$

where $\vec{V}_{\infty}$ is the undisturbed velocity given by:

$$\vec{V}_{\infty} = V_{\infty} (\cos \alpha \vec{i} + \sin \alpha \vec{j}) \quad (4.4)$$

and $\vec{v}_i$ is the induced velocity due to the potential flow singularities (sources and vortices) located on the panels. Each of the panels is represented by sources and vortices distributed along the length of the panel. The unit vectors of the velocities induced at any point (x', y') in the flowfield by source and vortex sheets located on an infinitesimally-small element ds_j of the j -th panel are denoted as $\vec{v}_\sigma$ and $\vec{v}_\gamma$, respectively. Therefore, the induced velocity field, $\vec{v}_i$, created by the source and vortex sheets located on the j -th panel, is expressed as

$$\vec{v}_i(x', y') = \int_0^{\Delta s_j} \vec{v}_\sigma \sigma_j(s) ds + \int_0^{\Delta s_j} \vec{v}_\gamma \gamma_j(s) ds \quad (4.5)$$

where $\sigma_j(s)$ and $\gamma_j(s)$ are the strength distributions of the source and vortex sheets along the panel, respectively, and Δs_j is the surface length of panel j .

4.2.1.2 Boundary Conditions

The first boundary condition requires the surface of the airfoil to be a streamline of the flow so that the local flow is tangent to the surface at all points along the surface. This condition is satisfied numerically by setting the normal component of the resultant

velocity at the control points of each panel to zero. If $V_{m_i}^s$ and $V_{m_i}^n$ are respectively the tangential and normal components of the velocity vector at the control point of the i -th panel, the boundary condition requires

$$V_{m_i}^n = 0, \quad i = 1, 2, \dots, N_{pan} \quad (4.6)$$

The normal and tangential velocity components are expressed through the superposition of the induced and undisturbed freestream velocity contributions as:

$$V_{m_i}^n = \sum_{j=1}^{N_{pan}} A_{i,j}^n \sigma_j + \sum_{j=1}^{N_{pan}} B_{i,j}^n \gamma_j + V_{\infty} \sin(\alpha - \theta_i) \quad (4.7)$$

$$V_{m_i}^s = \sum_{j=1}^{N_{pan}} A_{i,j}^s \sigma_j + \sum_{j=1}^{N_{pan}} B_{i,j}^s \gamma_j + V_{\infty} \cos(\alpha - \theta_i) \quad (4.8)$$

where $A_{i,j}^n$, $B_{i,j}^n$, $A_{i,j}^s$, $B_{i,j}^s$ are called influence coefficients. The source and vorticity strengths, σ and γ , respectively, appearing in the summation signs in Eqns. (4.7) and (4.8) play a significant role in determining the induced velocity magnitude. The influence coefficients are expressed as:

$$A_{i,j}^n = \begin{cases} \frac{1}{2\pi} \left[\sin(\theta_i - \theta_j) \ln \frac{r_{i,j+1}}{r_{i,j}} + \cos(\theta_i - \theta_j) \beta_{i,j} \right] & i \neq j \\ \frac{1}{2} & i = j \end{cases} \quad (4.9)$$

$$A_{i,j}^s = \begin{cases} \frac{1}{2\pi} \left[\sin(\theta_i - \theta_j) \beta_{i,j} - \cos(\theta_i - \theta_j) \ln \frac{r_{i,j+1}}{r_{i,j}} \right] & i \neq j \\ \frac{1}{2} & i = j \end{cases} \quad (4.10)$$

$$B_{i,j}^n = -A_{i,j}^s \quad (4.11)$$

$$B_{i,j}^s = A_{i,j}^n \quad (4.12)$$

where

$$r_{i,j+1} = \left[(x'_{m_i} - x'_{j+1})^2 + (y'_{m_i} - y'_{j+1})^2 \right]^{1/2} \quad (4.13)$$

$$r_{i,j} = \left[(x'_{m_i} - x'_j)^2 + (y'_{m_i} - y'_j)^2 \right]^{1/2} \quad (4.14)$$

$$x'_{m_i} = \frac{1}{2}(x'_i + x'_{i+1}), \quad y'_{m_i} = \frac{1}{2}(y'_i + y'_{i+1}) \quad (4.15)$$

$$\theta_i = \tan^{-1} \left(\frac{y'_{i+1} - y'_i}{x'_{i+1} - x'_i} \right), \quad \theta_j = \tan^{-1} \left(\frac{y'_{j+1} - y'_j}{x'_{j+1} - x'_j} \right) \quad (4.16)$$

$$\beta_{i,j} = \tan^{-1} \left(\frac{y'_{m_i} - y'_{j+1}}{x'_{m_i} - x'_{j+1}} \right) - \tan^{-1} \left(\frac{y'_{m_i} - y'_j}{x'_{m_i} - x'_j} \right) \quad (4.17)$$

in which $\beta_{i,j}$ is the angle between the two lines connecting the control point of the i -th panel, (x'_{m_i}, y'_{m_i}) , to the boundary points of the j -th panel, (x'_j, y'_j) and (x'_{j+1}, y'_{j+1}) , as shown in Figure 4.1.

The flow tangency boundary condition is insufficient to achieve a unique solution for the inviscid flowfield around the airfoil because only one single value of the circulation around the airfoil ensures that the flowfield at the airfoil trailing edge is physically realistic. To obtain the correct circulation value for airfoils with sharp trailing edges, the Kutta condition is applied (Cebeci et al., 2005), which states that for the inviscid flow to leave the trailing edge smoothly, the rear stagnation point should be exactly at the trailing edge. Cebeci et al. (2005) indicate that the most widely used approach to implement the Kutta condition is setting the upper and lower surface velocities to approach a common limit at the trailing edge. This limit is zero for airfoils with a nonzero angle between the two panels (panel 1 and panel N_{pan}) that form the

trailing edge. To implement this condition, the resultant tangential velocities at the two trailing edge panels are set to be equal:

$$\left| V_{m_{N_{pan}}}^s \right| = \left| V_{m_1}^s \right| \quad (4.18)$$

The two boundary conditions are satisfied by adjusting the strength of the source and vortex sheet on the panels, therefore making the source and vortex sheet strengths the primary unknowns in the problem. In the approach presented by Hess & Smith (1967), the source strength $\sigma(s)$ is constant over the j -th panel, and its value is adjusted on each of the panels to give zero normal velocity over the surface of the airfoil. The vorticity strength, γ , is assumed to be constant on all panels to satisfy the Kutta condition.

To get the final working equations, the flow tangency condition in Eqn. (4.6) is substituted into Eqn. (4.7):

$$\sum_{j=1}^{N_{pan}} A_{i,j}^n \sigma_j + \gamma \sum_{j=1}^{N_{pan}} B_{i,j}^n + V_{\infty} \sin(\alpha - \theta_i) = 0, \quad i = 1, 2, \dots, N_{pan} \quad (4.19)$$

The unknowns for the problem are the source strengths at the control points of N_{pan} number of panels and the uniform vortex-sheet strength, giving a total of $N_{pan} + 1$ unknowns. If Eqn. (4.19) is written for each panel individually, it provides N_{pan} equations. The $(N_{pan} + 1)$ -th equation is the Kutta condition (Eqn. (4.18)). These algebraic equations are formed into a system of equations, which is solved by the Gaussian elimination method.

4.2.1.3 System of Equations

If Eqns. (4.18) and (4.19) are formed into a set of algebraic equations in matrix form, the result is in the form of:

$$A\bar{\mathbf{x}} = \bar{\mathbf{b}} \quad (4.20)$$

where A is a square matrix of order $(N_{pan} + 1)$ consisting of influence coefficients:

$$A = \begin{bmatrix} a_{1,1} & a_{1,2} & \cdots & a_{1,j} & \cdots & a_{1,N_{pan}} & a_{1,N_{pan}+1} \\ a_{2,1} & a_{2,2} & \cdots & a_{2,j} & \cdots & a_{2,N_{pan}} & a_{2,N_{pan}+1} \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ a_{i,1} & a_{i,2} & \cdots & a_{i,j} & \cdots & a_{i,N_{pan}} & a_{i,N_{pan}+1} \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ a_{N_{pan},1} & a_{N_{pan},2} & \cdots & a_{N_{pan},j} & \cdots & a_{N_{pan},N_{pan}} & a_{N_{pan},N_{pan}+1} \\ a_{N_{pan}+1,1} & a_{N_{pan}+1,2} & \cdots & a_{N_{pan}+1,j} & \cdots & a_{N_{pan}+1,N_{pan}} & a_{N_{pan}+1,N_{pan}+1} \end{bmatrix} \quad (4.21)$$

In Eqn. (4.20), $\bar{\mathbf{x}} = (\sigma_1, \dots, \sigma_i, \dots, \sigma_{N_{pan}}, \gamma)^T$ and $\bar{\mathbf{b}} = (b_1, \dots, b_i, \dots, b_{N_{pan}}, b_{N_{pan}+1})^T$ with T

denoting the transpose. The elements of the coefficient matrix A , from Eqn. (4.19), are:

$$a_{i,j} = A_{i,j}^n, \quad i = 1, 2, \dots, N_{pan} \quad \text{and} \quad j = 1, 2, \dots, N_{pan} \quad (4.22)$$

$$a_{i,N_{pan}+1} = \sum_{j=1}^{N_{pan}} B_{i,j}^n, \quad i = 1, 2, \dots, N_{pan} \quad (4.23)$$

in which $A_{i,j}^n$ is calculated using Eqn. (4.9) and $B_{i,j}^n$ from Eqn. (4.11).

To calculate $a_{N_{pan}+1,j}$ ($j = 1, 2, \dots, N_{pan}$) and $a_{N_{pan}+1,N_{pan}+1}$ in matrix A , the tangential velocities in the Kutta condition (Eqn. (4.18)) are substituted in their open form given in Eqn. (4.8) while keeping the constant vortex-sheet strength, γ , out of the summation:

$$\sum_{j=1}^{N_{pan}} A_{1,j}^s \sigma_j + \gamma \sum_{j=1}^{N_{pan}} B_{1,j}^s + V_{\infty_r} \cos(\alpha - \theta_1) = - \left[\sum_{j=1}^{N_{pan}} A_{N_{pan},j}^s \sigma_j + \gamma \sum_{j=1}^{N_{pan}} B_{N_{pan},j}^s + V_{\infty_r} \cos(\alpha - \theta_{N_{pan}}) \right] \quad (4.24)$$

When Eqn. (4.24) is rearranged, the result is:

$$\sum_{j=1}^{N_{pan}} (A_{1,j}^s + A_{N_{pan},j}^s) \sigma_j + \gamma \sum_{j=1}^{N_{pan}} (B_{1,j}^s + B_{N_{pan},j}^s) = -V_{\infty_r} \cos(\alpha - \theta_1) - V_{\infty_r} \cos(\alpha - \theta_{N_{pan}}) \quad (4.25)$$

so that,

$$a_{N_{pan}+1,j} = A_{1,j}^s + A_{N_{pan},j}^s, \quad j = 1, 2, \dots, N_{pan} \quad (4.26)$$

$$a_{N_{pan}+1,N_{pan}+1} = \sum_{i=1}^{N_{pan}} (B_{1,i}^s + B_{N_{pan},i}^s) \quad (4.27)$$

in which $A_{1,j}^s$ and $A_{N_{pan},j}^s$ are calculated from Eqn. (4.10), and $B_{1,j}^s$ and $B_{N_{pan},j}^s$ from Eqn. (4.11).

The components of $\tilde{\mathbf{b}}$ are calculated using Eqns. (4.19) and (4.25):

$$b_i = -V_{\infty_r} \sin(\alpha - \theta_i), \quad i = 1, 2, \dots, N_{pan} \quad (4.28)$$

$$b_{N_{pan}+1} = -V_{\infty_r} \cos(\alpha - \theta_1) - V_{\infty_r} \cos(\alpha - \theta_{N_{pan}}) \quad (4.29)$$

Once all the elements of Eqn. (4.20) are determined, the system of equations is solved for the unknown vector $\tilde{\mathbf{x}}$, which contains the unknown source and vortex strengths. In the present algorithm, this operation is carried out using MATLAB[®]'s matrix division algorithm, which first attempts a Cholesky factorization (Press et al., 1988). If this approach fails, the coefficient matrix is factorized using Gaussian elimination if the matrix is upper Hessenberg (Datta, 2010), or an LU factorization with

partial pivoting (Datta, 2010) is used, if it is not. After the source and vortex strengths are determined, the inviscid velocity distribution over the airfoil surface is calculated.

4.2.1.4 Inviscid Flow Velocity and Pressure Coefficient Distribution

Once the source and vortex strengths are calculated, the velocity at the control point of each panel is determined. Since the velocity normal to the panels is zero at the control point, the tangential velocity, which is consequently the total velocity, is calculated from Eqn. (4.8). The inviscid velocity at the control point of each panel, $V_{m_i}^s$, is used to calculate the pressure coefficient, C_{p_i} , at the corresponding locations. The pressure coefficient, C_p , is defined as:

$$C_p = \frac{p - p_\infty}{\frac{1}{2} \rho V_{\infty}^2} \quad (4.30)$$

where p is the pressure at a given location, ρ is the fluid density, and p_∞ and V_∞ are the static pressure and velocity of the undisturbed freestream flow approaching the airfoil, respectively. The pressure coefficient can also be expressed in terms of the surface velocity as follows:

$$C_{p_i} = 1 - \left(\frac{V_{m_i}^s}{V_{\infty}} \right)^2, \quad i = 1, 2, \dots, N_{pan} \quad (4.31)$$

A maximum C_p value of 1.0 indicates a flow velocity of zero, i.e., a stagnation point.

Fluctuations are observed in the chordwise C_p distribution when using certain airfoil coordinate sets with significant discretization error. These fluctuations, if not cancelled out via numerical smoothing, get amplified over subsequent iterations of the

viscous-inviscid interactive solution due to the coupling between the viscous and inviscid flow solutions. Therefore, numerical smoothing is applied to the chordwise C_p distribution downstream of $x'/c = 0.15$, where c is the chord length, using MATLAB®'s 'smooth' function, which uses a moving average with a range of 7 panels. Smoothing is not performed close to the leading edge to ensure that the C_p predictions around the suction peak and the stagnation point are not modified.

4.2.1.5 Sizing of Airfoil Surface Panels

The analyses conducted in the present study focus on two airfoils: the NREL S809 and the NACA 0012. The S809 airfoil is a thick and cambered low-speed airfoil designed specifically for wind turbine blades, whereas NACA 0012 is a well-known, symmetric, and thin airfoil used in low-speed applications (see Appendix B for airfoil geometry and coordinates).

Airfoil coordinates are typically given with 30 to 50 coordinate points for each of the upper and lower surfaces of the airfoil. In a sensitivity analysis conducted for the NACA 0012 and S809 airfoils, two computations with different panel distributions were compared. The first set of computations used the original input coordinates as the boundary points, therefore the number of panels were controlled by the number of input coordinate points. The input coordinate spacing for the NACA 0012 airfoil was more refined at the leading edge, gradually increasing towards the trailing edge, as the streamwise surface curvature decreases. For the S809 airfoil, the input coordinates were more refined at both the leading edge and trailing edge, in keeping with more significant changes in surface curvature in those regions of the airfoil. In total, the NACA 0012 and

S809 airfoils were represented by a total of 100 and 60 panels, respectively. For the second set of computations, the numbers of panels on each airfoil were doubled, with the additional boundary point coordinates being calculated using a piecewise-cubic Hermite interpolating polynomial, which produces a smooth curve fit with no overshoot.

Both airfoils were tested at 0° and 10° angles of attack using the two different panel distributions. The difference between the predictions was no more than 2% of V_∞ , with the largest difference occurring close to the trailing edge. The predicted velocity differences on other parts of the airfoil were less than 1% of the reference velocity. However, it was observed that increasing the number of panels provides better resolution of the leading-edge stagnation point, as the panel boundary point near the leading edge with the lowest velocity is assumed to be the stagnation point. A precise location for the stagnation point is important for the accuracy of the viscous flow analysis.

Hess & Smith (1967) recommend that the size of a panel not be more than 50% greater than the adjacent panels. In addition, by comparing the solutions of the present panel method for a 2D cylinder in a cross-flow with the corresponding analytical solution, it was determined in the present study that the panel-angle difference between adjacent panels should not be more than 10° . It may be concluded that 120 panels are sufficient for the present inviscid flow analysis to obtain an accurate solution of the inviscid velocity and pressure fields and to identify the location of the stagnation point with sufficient resolution, provided the panel distribution is sufficiently refined in high curvature regions and the above-noted panel size and angle guidelines are followed.

4.2.2 Viscous Flow Analysis Using the Momentum-Integral Relation

The viscous flow analysis focuses on the boundary layer, where the viscous effects in the fluid are significant. Background information is provided in Section 2.5.1 for readers who are not familiar with the fundamentals of boundary layers. Among the methods available for boundary-layer analysis, solving the momentum-integral relation was chosen since it provides all the flow parameters that are deemed relevant to the aerodynamic performance of an airfoil in a fast and accurate manner, as discussed in the literature review.

4.2.2.1 The Momentum-Integral Relation

The momentum-integral relation for laminar and turbulent flows, as derived in detail in Section 2.5.2, is:

$$\frac{C_f}{2} = \frac{d\theta}{ds} + (2+H) \frac{\theta}{U_e} \frac{dU_e}{ds} \quad (4.32)$$

in which C_f is the skin friction coefficient, θ is the momentum thickness, H is the shape factor, U_e is the local freestream velocity at the edge of the boundary layer, and s is the streamwise coordinate along the length of the surface. Since this equation has three unknowns (C_f, θ, H) , two closure equations are required to obtain a unique solution. The methods to obtain the closure equations for laminar and turbulent boundary-layers are presented in Sections 2.5.2.1 and 2.5.2.2, respectively, and the corresponding laminar and turbulent momentum-integral calculation methods are explained in Sections 4.2.2.3 and 4.2.2.4, respectively.

Figure 4.2: Discretization of the streamwise distance along the upper and lower surfaces for the momentum-integral analysis.

The momentum-integral equation is integrated separately on the pressure and suction sides of the airfoil, from the stagnation point at the leading edge to the trailing edge. The discretization of the equation is performed on the basis of a series of computational nodes distributed along these lengths, as shown in Figure 4.2. More information on the solution grid of the momentum-integral analysis is presented in Section 4.2.2.2. The freestream velocity distribution over the surface, $U_e(s)$, is determined by interpolating the inviscid flow velocity, $V^s(s)$, calculated at the control points of each panel in the inviscid flow analysis, to the viscous flow computation nodes. In the present algorithm, a piecewise-cubic Hermite interpolating polynomial, which provides a smooth curve fit with no overshoot, is used for this process. Also the freestream velocity derivative, $(dU_e/ds)_i$, is approximated by using second-order differencing:

$$\left(\frac{dU_e}{ds}\right)_i \approx \begin{cases} \frac{U_{e_{i+1}} - U_{e_{i-1}}}{2\Delta s} & 2 \leq i \leq N_{vis} - 1 \\ \frac{3U_{e_i} - 4U_{e_{i-1}} + U_{e_{i-2}}}{2\Delta s} & i = N_{vis} \end{cases} \quad (4.33)$$

in which i is the node number and N_{vis} is the total number of nodes on each side of the airfoil. A second-order differencing is deemed sufficient, as the change in the calculated velocity differentials is negligible when a higher order differencing method is used.

4.2.2.2 Solution Grid for the Viscous Flow Analysis

The two solution grids for the viscous flow analysis on each side of the airfoil extend from the leading-edge stagnation point to the trailing edge. To perform momentum-integral calculations along the airfoil surface, uniform spacing has been used between the nodes, since it provides ease of programming. A grid sensitivity analysis was performed using different node spacing values to investigate whether uniform spacing is sufficient to obtain a grid-independent solution of the boundary-layer parameters in all regions of flow, including the areas with high rates of acceleration near the leading edge.

The grid sensitivity analysis was performed for the NACA 0012 and the S809 airfoils at 0° and 10° angles of attack, where node spacings ranging from 0.1% to 5% of the flow length on each side of the airfoil were tested. The skin-friction coefficient and the momentum thickness were used to assess grid sensitivity. It was observed that the change in these parameters was small when the grid was refined beyond a node spacing of 0.5%, such that the largest difference in these parameters for 0.5% and 0.2% spacings was about 2%. Therefore, it was determined that sufficiently accurate predictions could be achieved with uniform node spacing of 0.5% of the flow distance on each side of the airfoil. With this node spacing, the flow length on each side of the airfoil is divided into 200 sections, creating 201 nodes.

4.2.2.3 Laminar Momentum-Integral Calculation

The laminar momentum-integral calculation starts at the leading-edge stagnation point, continuing downstream past the point of transition inception, until the flow is fully turbulent, or flow separation is predicted. The momentum-integral equation for laminar boundary layers is iteratively solved at each node starting from the second node from the leading edge ($i=2$), since $i=1$ is taken to be the stagnation point where $C_f=0$ and $\theta=0$. The discretized form of the laminar momentum-integral relation as per the closure method of Thwaites (1949), described in Section 2.5.2.1, is:

$$\frac{S \left\{ \frac{(\theta_i^{n-1})^2}{\nu} \left(\frac{dU_e}{ds} \right)_i \right\} \nu}{\theta_i^{n-1} U_{e_i}} = \frac{3 \left(\frac{\theta_i^n - (1-\omega)\theta_i^{n-1}}{\omega} \right) - 4\theta_{i-1} + \theta_{i-2}}{2\Delta s} + \left(2 + H \left\{ \frac{(\theta_i^{n-1})^2}{\nu} \left(\frac{dU_e}{ds} \right)_i \right\} \right) \frac{\theta_i^{n-1}}{U_{e_i}} \left(\frac{dU_e}{ds} \right)_i \quad (4.34)$$

in which the subscripts with i indicate the node number, the superscripts with n indicate the iteration number, ω is an under-relaxation factor, and $\{ \}$ indicates the functions $S(\lambda)$ and $H(\lambda)$ given in Eqns. (2.66) and (2.67), respectively. The term $(d\theta/ds)$ in the momentum-integral relation (Eqn. (4.32)) is expressed in Eqn. (4.34) using a second-order backwards differencing approximation, since no downstream information on θ is available. The under-relaxation factor, ω , is set to 0.1 to ensure a numerically stable convergence. The convergence criteria can be adjusted by the user and by default, convergence is assumed to be reached when $|\theta_i^n - \theta_i^{n-1}| < \theta_i^{n-1} \times 10^{-5}$. If Eqn. (4.34) is rearranged for the target variable, θ_i^n :

$$\begin{aligned}
\theta_i^n = & \frac{2S \left\{ \frac{(\theta_i^{n-1})^2}{\nu} \left(\frac{dU_e}{ds} \right)_i \right\} \nu \Delta s \omega}{3\theta_i^{n-1} U_{e_i}} \\
& - \frac{2}{3} \omega \Delta s \left(2 + H \left\{ \frac{(\theta_i^{n-1})^2}{\nu} \left(\frac{dU_e}{ds} \right)_i \right\} \right) \frac{\theta_i^{n-1}}{U_{e_i}} \left(\frac{dU_e}{ds} \right)_i \\
& + \frac{4}{3} \omega \theta_{i-1} - \frac{1}{3} \omega \theta_{i-2} + (1 - \omega) \theta_i^{n-1}
\end{aligned} \tag{4.35}$$

If $\theta_2^0 = \theta_1 = 0$ is assumed for the initialization of the calculation at node 2, it can be seen from Eqn. (4.35) that the solution runs into a singularity due to the term θ_i^{n-1} in the denominator. To circumvent this problem, a non-zero initial estimation is needed for θ_2^0 . To get this estimation, θ_2^0 is expressed using first-order backward differencing, and the under-relaxation factor is dropped:

$$\left(\frac{d\theta}{ds} \right)_2^0 \approx \frac{\theta_2^0 - \theta_1}{\Delta s} = \frac{\theta_2^0}{\Delta s} \tag{4.36}$$

If the momentum-integral relation (Eqn. (4.32)) for laminar boundary layers is rewritten at $i = 2$ without the under-relaxation factor, and a zero pressure gradient is assumed as a first-order approximation, after rearranging:

$$\theta_2^0 = \frac{S\{0\} \nu}{\left(\frac{d\theta}{ds} \right)_2^0 U_{e_2}} \tag{4.37}$$

If Eqn. (4.36) is substituted into Eqn. (4.37), noting that $S(0) = 0.22$,

$$\theta_2^0 = \sqrt{\frac{\Delta s 0.22 \nu}{U_{e_2}}} \tag{4.38}$$

Eqn. (4.38) is used to obtain a reasonable first guess for θ_2 . In subsequent iterations ($n \geq 2$), θ_2 converges to its actual value, which includes the effect of the local streamwise pressure gradient.

Thwaites' correlations are valid for the range of $-0.09 \leq \lambda \leq 0.25$. In the present algorithm, for $\lambda > 0.25$ (i.e., highly-favourable pressure gradients), the S and H values corresponding to $\lambda = 0.25$ are retained, and for $\lambda < -0.09$ (incipient separation), the algorithm changes the calculation routine to one that corresponds to a separated flow, as presented in Section 4.2.2.7.

4.2.2.4 Turbulent Momentum-Integral Calculation

The turbulent momentum-integral calculation starts at the transition inception location, which is where turbulent spots start developing in the boundary layer (Emmons, 1951), and continues downstream. At the location of transition onset, however, the momentum thickness for the turbulent momentum-integral calculation is zero, analogous to node $i = 1$ being the stagnation point for the laminar momentum-integral calculation. Therefore, the calculation starts at the neighbouring downstream node, $i^* = i_{ts} + 1$, in which i_{ts} is the node of transition inception. The discretized form of the turbulent momentum-integral relation as per the closure method suggested by White (2006), described in Section 2.5.2.2, is:

$$\frac{\frac{0.3e^{-1.33H_i^n}}{(\log_{10}(\text{Re}_\theta)_i^{n-1})^{1.74+0.31H_i^n}} \omega_2 + (1-\omega_2)C_{fi}^{n-1}}{2} = \frac{3 \left(\frac{\theta_i^n - (1-\omega_1)\theta_i^{n-1}}{\omega_1} \right) - 4\theta_{i-1} + \theta_{i-2}}{2\Delta s} + (2 + H_i^n) \frac{\theta_i^{n-1}}{U_{e_i}} \left(\frac{dU_e}{ds} \right)_i \quad (4.39)$$

in which

$$H_i^n = \frac{\sqrt{\frac{2}{C_{f_i}^{n-1}}}}{\sqrt{\frac{2}{C_{f_i}^{n-1}} + a\{\Pi_i^n\}}} \quad (4.40)$$

and

$$a\{\Pi_i^n\} = \frac{2 + 3.179\Pi_i^n + 1.5(\Pi_i^n)^2}{\kappa(1 + \Pi_i^n)} \quad (4.41)$$

where Π_i^n is calculated by solving:

$$-0.4 + 0.76\Pi_i^n + 0.42(\Pi_i^n)^2 = -\frac{2}{C_{f_i}^{n-1}} H_i^{n-1} \frac{\theta_i^{n-1}}{U_{e_i}} \left(\frac{dU_e}{ds} \right)_i \quad (4.42)$$

In the above equations, Re_θ is the Reynolds number ($Re_\theta = U_e \theta / \nu$) based on momentum thickness, Π is Coles' wake parameter, ω_1 and ω_2 are under-relaxation factors for θ and C_f , respectively, the subscript i indicates the node number, and the superscript n indicates the iteration number. MATLAB[®]'s 'roots' function, which computes the roots of a polynomial equation by calculating the eigenvalues of its companion matrix, is used to solve Eqn. (4.42). The under-relaxation factors, ω_1 and ω_2 are both set to 0.5 to ensure a numerically stable convergence. Convergence is assumed to be reached when $|\theta_i^n - \theta_i^{n-1}| < \theta_i^{n-1} \times 10^{-5}$ and $|C_{f_i}^n - C_{f_i}^{n-1}| < C_{f_i}^{n-1} \times 10^{-5}$. If Eqn. (4.39) is rearranged for θ_i^n :

$$\theta_i^n = \frac{\Delta s}{3} \omega_1 \left(\frac{0.3e^{-1.33H_i^n}}{\left(\log_{10}(\text{Re}_\theta)_i \right)^{n-1}} \omega_2 + (1 - \omega_2) C_{f_i}^{n-1} \right) - \frac{2\Delta s}{3} \omega_1 (2 + H_i^n) \frac{\theta_i^{n-1}}{U_{e_i}} \left(\frac{dU_e}{ds} \right)_i + \frac{4}{3} \omega_1 \theta_{i-1} - \frac{1}{3} \omega_1 \theta_{i-2} + (1 - \omega_1) \theta_i^{n-1} \quad (4.43)$$

White (2006) indicates that two of the three unknowns of the momentum-integral relation $(H_i, \text{ and } C_{f_i}, \text{ for example})$ need to be known at the first calculation node, i^* , to initialize the turbulent momentum-integral calculation. If two of the three unknowns at node i^* are known, the third unknown $(\theta_i, \text{ in this example})$ is directly determined from the turbulent momentum-integral relation, therefore it is not possible to iteratively solve the closure equations at node i^* . What follows is the method used to initialize the turbulent momentum-integral calculation using an approximate solution at node i^* . First, $(d\theta/ds)_i$ is expressed using first-order backwards differencing:

$$\left(\frac{d\theta}{ds} \right)_i \approx \frac{\theta_i - \theta_{i-1}}{\Delta s} = \frac{\theta_i}{\Delta s} \quad (4.44)$$

where $\theta_{i-1} = 0$ since the momentum thickness of the turbulent boundary layer at the transition inception location is assumed to be zero. The momentum-integral relation is then simplified by assuming a zero-pressure gradient:

$$\frac{C_{f_i}}{2} = \left(\frac{d\theta}{ds} \right)_i \quad (4.45)$$

If Eqns. (4.44) and (4.45) are substituted into Eqn. (2.74), the following is obtained:

$$C_{f_i^*} \approx \frac{0.3e^{-1.33H_i^*}}{\left(\log_{10} \frac{\Delta s C_{f_i^*} U_{e_i^*}}{2\nu} \right)^{1.74+0.31H_i^*}} \quad (4.46)$$

Eqn. (4.46) is solved for $C_{f_i^*}$ by assuming $H_i^* = 1.5$, since the shape factor value at upstream locations of a turbulent boundary layer is usually large due the decrease in H from higher laminar boundary-layer values. No significant change in the value of $C_{f_i^*}$ is observed when greater H_i^* values are assumed. In the present algorithm, Eqn. (4.46) is solved using MATLAB[®]'s 'fsolve' function, which uses a trust-region dogleg algorithm (Powell, 1968). Once $C_{f_i^*}$ is calculated, θ_i^* is obtained from

$$\theta_i^* \approx \frac{\Delta s C_{f_i^*}}{2} \quad (4.47)$$

The iterative calculation is then performed at node $i^* + 1$ and downstream using Eqns. (4.42), (4.41), (4.40), and (4.43), with $\theta_{i^*+1}^0 = \theta_i^*$, $C_{f_{i^*+1}}^0 = C_{f_i^*}$, and $H_{i^*+1}^0 = H_i^*$.

Turbulent boundary-layer separation is predicted when the shape factor exceeds $H = 2.5$ (White, 2006), or if the value of the skin-friction coefficient drops below $C_f = 0$. In such a case, the turbulent momentum-integral calculation is terminated and the algorithm changes to the calculation routine for separated flow, presented in Section 4.2.2.7.

4.2.2.5 Transition Model

Since the momentum-integral equation can only be solved for either laminar or turbulent flows, a transition model is necessary to calculate the boundary-layer

parameters in the transition region. Available models to predict the mode of transition (natural, bypass, or separation-bubble), transition inception and completion locations, and the transition rate are discussed in Section 3.3.3.3. Among these available methods, the model provided by Roberts & Yaras (2005b) is used in the present algorithm. The transition model uses Abu-Ghannam & Shaw's (1980) correlation to predict the transition inception location for attached boundary layers:

$$\text{Re}_{\theta_{ts}} = 163 + \exp\left(F(\lambda) - \frac{F(\lambda)}{6.91} \overline{Tu}\right) \quad (4.48)$$

where $\text{Re}_{\theta_{ts}}$ is the Reynolds number based on the momentum thickness and freestream velocity at the transition inception location; $\overline{Tu}$ is the average freestream turbulence intensity (expressed as a percentage) occurring between the leading edge and the transition inception location; and $F(\lambda)$ is a function of Thwaites' pressure gradient parameter, λ (Eqn. (2.64)), defined as:

$$F(\lambda) = \begin{cases} 6.91 + 12.75\lambda + 63.64\lambda^2 & \lambda \leq 0 \\ 6.91 + 2.48\lambda - 12.27\lambda^2 & \lambda \geq 0 \end{cases} \quad (4.49)$$

If the flow separates before $\text{Re}_{\theta_{ts}}$ is reached, separation-bubble transition is assumed to take place. In such a case, Roberts & Yaras' (Roberts & Yaras, 2005b) correlation to predict the transition inception location is used, without the effects of surface roughness (since the momentum-integral method does not include the effects of surface roughness):

$$\text{Re}_{sep-ts} = (835 - 36TF(\%)) \text{Re}_{\theta_{sep}}^{0.7} \quad (4.50)$$

where Re_{sep-ts} is the Reynolds number based on the streamwise distance between the separation and transition inception locations and $\text{Re}_{\theta_{sep}}$ is the Reynolds number based on

the momentum thickness and freestream velocity at the separation point. TF is Taylor's turbulence factor, defined as:

$$TF = Tu_{ref} (L/\lambda_s) \quad (4.51)$$

where Tu_{ref} is the reference turbulence intensity (expressed as a percentage) 10 mm upstream of the leading edge, assumed to be equal to $\overline{Tu}$; L is the flow surface length; and λ_s is the integral length scale of turbulence. The streamwise Reynolds number at the transition inception location is:

$$Re_{ts} = Re_{sep} + Re_{sep-ts} \quad (4.52)$$

where $Re_{sep} = U_e s_{sep} / \nu$ is the Reynolds number at the streamwise separation location, s_{sep} .

Once the transition inception location is known using Eqn. (4.48) or (4.52), the intermittency, γ , is calculated at each node within the transition region. The unified transition model of Roberts & Yaras (2005b) allows the intermittency for both attached and separated boundary layers to be calculated using:

$$\gamma = 1 - \exp \left(-n \int_{s_{ts}}^s \left[\sigma / \tan(\alpha) U_e \right] ds \int_{s_{ts}}^s \tan(\alpha) ds \right) \quad (4.53)$$

in which n is the turbulent spot production rate in $(ms)^{-1}$, σ is the spot propagation parameter, α is the spot-spreading-half-angle, and s_{ts} is the streamwise transition inception location. The spot production rate, n , is calculated from:

$$N = \frac{n \sigma \theta_{ts}^3}{\nu} \quad (4.54)$$

in which θ_{ts} is the momentum thickness at the transition inception location; and N is the spot inception rate, which is calculated from

$$\log_{10}(N) = \frac{0.55H_{ts} - 2.2}{1 - 0.63H_{ts} + 0.14H_{ts}^2}, \quad 1.6 \leq H_{ts} \leq 8.5 \quad (4.55)$$

where H_{ts} is the shape factor at the transition inception location. The spot propagation parameter, σ , and the spot-spreading-half-angle, α , are calculated using the following equations:

$$\sigma = 0.024 + \frac{0.604}{1.0 + 5.0e^{66.0\lambda}} \quad (4.56)$$

$$\alpha = 4.0 + \frac{22.14}{0.79 + 2.72e^{47.63\lambda}} \quad (4.57)$$

in which λ is Thwaites' pressure gradient parameter (Eqn. (2.64)). The momentum thickness distribution within the transition region is required to calculate λ . To the author's knowledge, no method for estimating θ in this region without knowing the transition completion location *a priori* has been published. Therefore, the following intermittency-based approximation is used to combine the information provided by the laminar and turbulent momentum-integral calculations to get a reasonable estimate for θ within the transition region:

$$(\theta)_i = \gamma_{i-1}\theta_{i,\text{turbulent}} + (1 - \gamma_{i-1})\theta_{i,\text{laminar}} \quad (4.58)$$

In Eqn. (4.58), the intermittency value at node $i-1$ is used since the intermittency at node i is not known. When using the recommended node spacing (0.5% of the flow distance), the change in the intermittency value between the two neighbouring nodes for attached-flow transition is sufficiently small ($\leq 5\%$) so that the error caused by using the intermittency value at the upstream node is negligible.

To implement the transition model to the present viscous flow analysis algorithm, at each node i , where the flow is fully laminar, the Reynolds number based on momentum thickness for attached-flow transition inception, Re_{θ_i} , is calculated using Eqn. (4.48). If $Re_{\theta_i} \geq Re_{\theta_{ts}}$, attached-flow transition is assumed to start at node $i = i_{ts}$, where i_{ts} is the node of transition inception. If flow separation is predicted by the laminar momentum-integral calculation at a node i where $Re_{\theta_i} < Re_{\theta_{ts}}$, the laminar momentum-integral calculation is terminated and separation-bubble transition is assumed to take place. In this case, the streamwise Reynolds number for transition inception, Re_{is} , is calculated using Eqns. (4.50), (4.51), and (4.52), and transition inception is assumed to take place at node $i = i_{ts}$, which satisfies $(Re_s)_i \geq Re_{is}$, where $(Re_s)_i$ is the streamwise Reynolds number at node i . Once i_{ts} is determined, the spot inception rate, N , is calculated using Eqn. (4.55) by using the shape factor at the transition inception location, H_{ts} . For separation-bubble transition, the last known shape factor value from the laminar momentum-integral calculation is assumed to be equal to H_{ts} , since transition inception usually occurs shortly after separation. The turbulent momentum-integral calculation is initialized at node $i^* = i_{ts} + 1$ and is continued at each downstream node. The intermittency function, γ , is calculated at every node starting at node i_{ts} using Eqns. (4.56), (4.57), (4.54), and (4.53) until $\gamma = 0.99$ is reached, where the boundary layer is assumed to be fully turbulent. It should be noted that the transition model assumes that flow reattachment occurs at the location of transition completion for separation-bubble transition (Roberts & Yaras, 2005b).

4.2.2.6 Calculation of the Boundary-Layer Parameters

The boundary-layer parameters between the stagnation point and the transition inception location are assumed to be equal to the predictions of the laminar momentum-integral calculation, since the flow in that region is fully laminar. The boundary-layer parameters in the transition region are calculated by using information provided by the transition model in conjunction with the predictions of the laminar and turbulent momentum-integral calculations. It should be noted that the predictions of the laminar and turbulent momentum-integral calculations at all nodes are kept independent from each other. The calculations for attached- and separated-flow transition are presented in Sections 4.2.2.6.1 and 4.2.2.6.2, respectively. At the end of the transition region, where the flow is considered fully turbulent, the laminar momentum-integral calculation is stopped and the algorithm continues only with the turbulent momentum-integral calculation. It must be ensured that the boundary-layer parameters calculated at the transition completion location are equal to the turbulent momentum-integral calculation predictions at the same location, so that any discontinuity is avoided. At all nodes downstream of the transition completion location, the boundary-layer parameters are assumed to be equal to the predictions of the turbulent momentum-integral calculation.

4.2.2.6.1 Calculation of the Boundary-Layer Parameters in an Attached-Flow Transition Region

Abu-Ghannam and Shaw (1980) propose the following empirical relation for the momentum thickness distribution in attached, transitional boundary layers:

$$\theta' = \eta^{1.35} \quad (4.59)$$

in which

$$\eta = \frac{s - s_{ts}}{s_{te} - s_{ts}} \quad (4.60)$$

and

$$\theta' = \frac{\theta(s) - \theta_{ts}}{\theta_{te} - \theta_{ts}} \quad (4.61)$$

where θ' is the momentum thickness ratio, η is the nondimensional distance in the transition region, s is the streamwise distance, and the subscripts ts and te indicate the transition onset and completion locations, respectively.

No method for calculating the displacement thickness distribution in the transitional region was found in published literature. Therefore, a third-order Bézier curve fit (Farin, 2002) between the values of the displacement thickness at the transition inception and completion locations is used to estimate the displacement thickness distribution in the transition region, since the values of the displacement thickness at these locations are known from the respective laminar and turbulent momentum-integral calculations. A Bezier curve is chosen because of its ability to match both the values and the local derivative at the two sides of the curve-fit region, and thus the overall displacement thickness distribution (including the laminar, transitional, and turbulent regions) is continuous up to the first derivative.

The skin friction coefficient in the transitional region is calculated using the intermittency function as follows:

$$(C_f)_i = \gamma_i (C_{f, \text{turbulent}})_i + (1 - \gamma_i) (C_{f, \text{laminar}})_i \quad (4.62)$$

where $(C_{f, \text{turbulent}})_i$ and $(C_{f, \text{laminar}})_i$ are respectively the skin friction coefficients obtained from the turbulent and laminar momentum-integral calculations at node i in the transitional region.

4.2.2.6.2 Calculation of the Boundary-Layer Parameters in a Separated-Flow Transition Region

As there is limited experimental data for separated flows, certain simplifications and assumptions are made to calculate the boundary-layer parameters within separation bubbles. The skin-friction coefficient, C_f , is assumed to be zero for separated flows, as the skin friction within separation bubbles is very low and the skin friction generated by recirculation zones is negative, and thus has a decreasing effect on the skin-friction drag (Gaster, 1969; Singh & Sarkar, 2011).

Another boundary-layer parameter that requires modelling within the separation bubble is the displacement thickness. To the author's knowledge, no correlation has been developed to predict the displacement thickness within separation bubbles, and therefore, the only information available in the literature is experimental data. Roberts (2005) published displacement thickness information for separation bubbles in flows with streamwise Reynolds numbers ranging from 3.5×10^5 to 4.7×10^5 , and freestream turbulence intensities between 0.4% and 4.5%, along with the corresponding separation, transition inception, and reattachment locations. This experimental data, summarized in Table 4.1, is used by the present algorithm to predict the displacement thickness within separation bubbles based on the nondimensional location of the maximum displacement

thickness in the separation bubble $\left(s_{\delta_{\max}^*} - s_{sep} \right) / \left(s_{te} - s_{sep} \right)$ and the ratio of the maximum displacement thickness to the displacement thickness at the separation location, $\delta_{\max}^* / \delta_{sep}^*$. A 2D linear interpolation to calculate these parameters is performed for flows with streamwise Reynolds numbers and turbulence intensities that lie within Roberts' test matrix, and the limit values are used for flows that lie outside the test matrix. Then, a second-order polynomial curve, assumed to represent displacement thickness distribution within the separation bubble, is calculated using s_{sep} , $\left(s_{\delta_{\max}^*} - s_{sep} \right) / \left(s_{te} - s_{sep} \right)$, and s_{te} as the independent variables and δ_{sep}^* , $\delta_{\max}^*$, and δ_{te}^* as the dependent variables at the corresponding streamwise locations. This approximation for the displacement thickness does not introduce significant errors to the calculation of the overall boundary-layer development over the full length of the airfoil due to the separation bubbles generally being small compared to the full chord length, especially at high Reynolds numbers and freestream turbulence intensities (Roberts, 2005).

Table 4.1: Experimental data used for estimating the displacement thickness within separation bubbles (Roberts, 2005).

Freestream turbulence intensity (%)	Re = 350,000		Re = 470,000	
	$\frac{\left(s_{\delta_{\max}^*} - s_{sep} \right)}{\left(s_{te} - s_{sep} \right)}$	$\delta_{\max}^* / \delta_{sep}^*$	$\frac{\left(s_{\delta_{\max}^*} - s_{sep} \right)}{\left(s_{te} - s_{sep} \right)}$	$\delta_{\max}^* / \delta_{sep}^*$
0.4	0.633	4.42	0.6	3.88
4.5	0.6	2.33	0.526	2.14

Experimental data on the momentum thickness distribution within a separation bubble has been published by Roberts & Yaras (2005a). However, the transition completion location must be known *a priori* to calculate the momentum thickness using a

curve fit approach similar to the one described for the displacement thickness, which is not possible because the momentum thickness information is required to calculate the intermittency and the transition completion location. Therefore, the momentum thickness distribution within the bubble is approximated by extrapolating from the momentum thickness (θ_{sep}) and momentum thickness growth rate $((d\theta/ds)_{sep})$ values at the separation location. Any errors introduced when calculating the boundary-layer parameters in the transitional region are not expected to significantly affect the 2D aerodynamic performance predictions. This is primarily because the length of the transition region and the size of separation bubbles is typically small in comparison to the fully-laminar or fully-turbulent regions of the flow over the airfoil.

4.2.2.7 Treatment of Flow Separation

At the point of flow separation, the boundary layer becomes a free shear layer and travels away from the airfoil surface, leaving a ‘dead’ zone of weak eddies that generate some recirculation between itself and the airfoil surface, while the freestream continues to flow above it (Katz & Plotkin, 1991). The algorithm of the present study treats flow separation differently depending on whether flow separation occurs in the fully-laminar, transitional, or the fully-turbulent flow state. If flow separation occurs in the fully-laminar flow state, the laminar momentum-integral calculation is stopped at the point of separation and separation-bubble transition is assumed to take place, as described in Section 4.2.2.5. The calculation of the boundary-layer parameters within the separation bubble is presented in Section 4.2.2.6.2. If transition inception does not occur within 30% chordwise distance from the separation location, the separated shear layer is assumed to

remain detached from the airfoil surface for the remainder of the streamwise surface length. The calculation of the boundary-layer parameters in such a case is explained later in this section. Since both laminar and turbulent momentum-integral calculations are performed within an attached-flow transition region, the algorithm treats the prediction of flow separation by the two momentum-integral calculations differently. If separation is predicted only by the laminar momentum-integral calculation within this region, the boundary layer is assumed to remain attached to the airfoil surface. In such a case, the transition model described in Section 4.2.2.5 is followed without any modification, and the boundary-layer parameters for the transitional region are calculated as described in Section 4.2.2.6.1. The skin-friction factor for the laminar momentum-integral calculation is assumed to stay constant at its last known value, and the momentum and displacement thicknesses predicted by the laminar momentum-integral calculation are assumed to keep increasing at their growth rates prevailing just upstream of the location of separation-prediction, until transition is complete. If separation is predicted by the turbulent momentum-integral calculation at a location within an attached-flow transition region, the boundary layer is assumed to separate at this location. In such a case, or if the boundary layer is in a fully-turbulent state at the point of separation, the flow is assumed not to reattach to the airfoil surface.

In cases of boundary-layer separation without downstream reattachment, the skin-friction coefficient is set to zero for the full length of the airfoil surface downstream of the point of separation. Conventionally, regions of separated flow have been treated by assuming a uniform pressure distribution between the separation point and the trailing edge (Katz & Plotkin, 1991). However, assuming a uniform pressure distribution after the

separation point does not provide the necessary displacement thickness information for the subsequent inviscid flow analysis. It is assumed that there is no net mass flow in the separated-flow zone, thus the displacement thickness is set equal to the distance of the separated shear layer from the airfoil surface. Accordingly, the displacement thickness after separation grows in proportion to the angle between the separated shear layer and the airfoil surface:

$$\frac{d\delta^*}{ds} = m \tan(\alpha - \theta) \quad (4.63)$$

where $d\delta^*/ds$ is the streamwise growth rate of the displacement thickness, α is the airfoil angle of attack, and θ is the angle between the airfoil surface and the x' axis (see Figure 4.3). The proportionality constant, m , is an empirical free-shear-layer angle

Figure 4.3: Schematic of a shear layer (thickness exaggerated) leaving the airfoil surface after flow separation.

coefficient. A value of $m = 0$ indicates that the separated shear layer follows a trajectory that is parallel to the surface of the airfoil, which may be applicable for low angles of attack. Conversely, $m = 1$ indicates that the free shear layer leaves the airfoil in the same direction as the undisturbed freestream flow, which is more likely for highly-separated

flows. Any value of m between 0 and 1 indicates that the separated shear is oriented at a direction between the undisturbed freestream flow direction and the airfoil surface.

Figure 4.4: Correlation between stalled angles of attack and tangent coefficient, m , for the (a) S809 and the (b) NACA 0012 airfoils.

Figure 4.5: Error of the predicted C_l at higher- and lower-than-optimum values of m for the (a) S809 and the (b) NACA 0012 airfoils.

In a sensitivity analysis, m values within the range of $0 \leq m \leq 1$ with increments of 0.025 were used to calculate the aerodynamic coefficients for the S809 and NACA 0012 airfoils for stalled angles of attack up to 20° . When the predicted lift coefficients

were compared to the results of 2D experiments under similar flow conditions, it was discovered that for stalled angles of attack, a close-to-linear correlation exists between α and the value of m that gives C_l prediction with the smallest error. This close-to-linear correlation is used in the development of the so-called “stalled-aerodynamics prediction method” of the present study. The correlations for the S809 and the NACA 0012 airfoils are shown in Figures 4.4a and 4.4b, respectively. The sensitivity of the predicted lift coefficients ($C_{l_{pred}}$) to small deviations from the optimum value of m are shown for the corresponding airfoils in Figures 4.5a and 4.5b, respectively. It can be seen in Figure 4.5 that small changes in the value of m do not have a significant effect on the predicted value of C_l until heavily-stalled angles of attack. The experimental lift coefficients ($C_{l_{exp}}$) that were used to develop the correlations are provided by Somers (1997) and Abbott & von Doenhoff (1959) for the S809 and NACA 0012 airfoils, at Reynolds numbers based on chord length, $Re_c = 2 \times 10^6$ and $Re_c = 3 \times 10^6$, respectively.

For an airfoil at a stalled angle of attack α , the value of m corresponding to α given by the correlation for the airfoil is used in Eqn. (4.63) to calculate the displacement thickness growth rate downstream of the flow separation location. For unstalled angles of attack, $m = 0$ is used. If the value of m calculated by the correlation exceeds 1, $m = 1$ is used, since this value corresponds to the maximum value that is physically possible.

4.2.3 Calculation of the 2D Aerodynamic Coefficients

To calculate the 2D lift and drag coefficients, first the lift and drag forces are resolved in the x' and y' coordinates that are fixed to the airfoil chord line (shown in

Figure 4.1). The skin-friction drag force is calculated using $C_f(s)$ to calculate the wall shear stress, $\tau_w(s)$, at every node using Eqn. (2.53). The wall shear stress is then integrated along the surfaces of the airfoil to find the x' and y' components of the skin-friction drag force:

$$(D_{skin})_{x'} = \int_0^L \tau_w(s) \cos[\theta(s)] ds \quad (4.64)$$

$$(D_{skin})_{y'} = \int_0^L \tau_w(s) \sin[\theta(s)] ds \quad (4.65)$$

The calculated drag force components are then normalized by the undisturbed freestream dynamic pressure and the chord length, c :

$$(C_{d_{skin}})_{x'} = \frac{(D_{skin})_{x'}}{\frac{1}{2} \rho V_{\infty}^2 c} \quad (4.66)$$

$$(C_{d_{skin}})_{y'} = \frac{(D_{skin})_{y'}}{\frac{1}{2} \rho V_{\infty}^2 c} \quad (4.67)$$

Noting the drag is always parallel to the undisturbed freestream flow direction, the total skin-friction drag coefficient is then calculated:

$$C_{d_{skin}} = (C_{d_{skin}})_{x'} \cos(\alpha) + (C_{d_{skin}})_{y'} \sin(\alpha) \quad (4.68)$$

To calculate the forces in the x' and y' directions created by the pressure distribution around the airfoil, the x' and y' components of the force on each inviscid flow analysis panel are calculated and summed:

$$C_{x'_{pres}} = \sum_{i=1}^{N_{pan}} C_{p_i} \frac{\Delta s_i}{c} \sin(\theta_i) \quad (4.69)$$

$$C_{y'_{pres}} = \sum_{i=1}^{N_{pan}} C_{p_i} \frac{\Delta s_i}{c} \cos(\theta_i) \quad (4.70)$$

$C_{x'_{pres}}$ and $C_{y'_{pres}}$ are the resultant force coefficients in the x' and y' directions, respectively, created by the pressure distribution; N_{pan} is the total number of panels; and Δs is the surface length of each panel. The pressure drag coefficient, $C_{d_{pres}}$, is then calculated as:

$$C_{d_{pres}} = C_{x'_{pres}} \cos(\alpha) + C_{y'_{pres}} \sin(\alpha) \quad (4.71)$$

The total force coefficients in the x' and y' directions, $C_{x'}$ and $C_{y'}$, are calculated by summing the forces created by the pressure distribution and skin friction in the corresponding directions:

$$C_{x'} = C_{x'_{pres}} + (C_{d_{skin}})_{x'} \quad (4.72)$$

$$C_{y'} = C_{y'_{pres}} + (C_{d_{skin}})_{y'} \quad (4.73)$$

Finally, the total lift and drag coefficients are calculated by resolving the force coefficients $C_{x'}$ and $C_{y'}$, in the lift and drag directions:

$$C_l = -C_{x'} \sin(\alpha) + C_{y'} \cos(\alpha) \quad (4.74)$$

$$C_d = C_{x'} \cos(\alpha) + C_{y'} \sin(\alpha) \quad (4.75)$$

The pitching-moment coefficient, C_m , is usually calculated about the aerodynamic centre of the airfoil. To calculate the pitching moment, the total force in the y' direction on each panel, which is assumed to act on the control point of each panel, is evaluated and the total moment about the aerodynamic centre due to these forces is calculated:

$$C_m = \sum_{i=1}^{N_{pan}} -C_{p_i} \frac{\Delta s_i}{c^2} \cos(\theta_i) (0.25c - x'_{m_i}) \quad (4.76)$$

or

$$C_m = \sum_{i=1}^{N_{pan}} C_{p_i} \frac{\Delta s_i}{c^2} \cos(\theta_i) (x'_{m_i} - 0.25c) \quad (4.77)$$

where x'_{m_i} is the x' coordinate of the control point of panel i and c is the chord length.

Here, the negative sign indicates a nose-down pitching moment, which is typical for airfoils at positive angles of attack in unstalled conditions.

4.2.4 Interfacing of the Viscous and Inviscid Flow Solutions

The calculation procedure for the 2D airfoil aerodynamics starts with an inviscid flow analysis, as described in Section 4.2.1. The predicted freestream velocity distribution is then used as input in the viscous flow analysis (Section 4.2.2), followed by the calculation of the aerodynamic coefficients (Section 4.2.3). The panels representing the airfoil surface are displaced outwards from the original panel coordinates by a distance equal to the predicted local displacement thickness to account for the presence of the boundary layer. Once the panels are displaced, a new inviscid flow analysis is conducted with the updated panel locations, followed by another viscous flow analysis. This interactive scheme continues until convergence to the desired level is reached. The solution is assumed to be converged if the absolute differences between the current and preceding values of both the lift and drag coefficients for the last two iterations differ by less than 0.001.

For low and moderate angles of attack ($\alpha < 10^\circ \sim 15^\circ$), convergence is usually reached within 30 iterations of the viscous-inviscid interactive calculations. At higher angles of attack, the chordwise location of turbulent separation on the suction side is noted to fluctuate between sequential viscous flow analyses, usually in a cyclic manner. This behaviour is noted to cause the aerodynamic coefficient values predicted between consecutive iterations of the interactive solution to fluctuate by about 5-10%, even though the mean value of the aerodynamic coefficients averaged over 8-10 iterations is noted to remain steady ($\leq 2\%$ difference between the aerodynamic coefficients predicted by averaging the results of different consecutive iteration sets) after the first 20 iterations. The calculation is limited to 50 iterations, and if this limit is reached, the values predicted in the last 20 iterations are averaged to dampen the effect of small fluctuations in the calculated aerodynamic coefficients.

To promote numerical stability and reduce the amplitude of fluctuations over iteration cycles in the aerodynamic coefficients observed for calculations involving high angles of attack, under-relaxation is applied to the displacement thickness distributions predicted during each iteration of the viscous flow analysis. The under-relaxation factor is set at 0.15, established by trial and error. With this value of the under-relaxation factor, at least 29 iterations are required for the displacement thickness to reach 99% of its converged value. A minimum of 20 iterations is also enforced, which is sufficient for δ^* to reach 96% of its converged value.

To illustrate how the different sections of the viscous-inviscid interaction method come together, a brief summary of the solution algorithm is presented here, and a flowchart of the algorithm is given in Appendix A.

The main inputs of the 2D aerodynamic analysis algorithm are the airfoil coordinates, angle of attack, Reynolds number based on chord length, freestream turbulence intensity, integral length scale of turbulence, and the correlation for the stalled-aerodynamics prediction method, if available. The algorithm proceeds in the following manner:

- 1) The airfoil coordinates are sorted to be used in the panel method; starting from the pressure side of the trailing edge up to the leading edge, and back to the trailing edge on the suction side. The coordinates are used directly as boundary points and no additional panels are created by the algorithm.
- 2) The original airfoil geometry, including the angle of each panel is stored independently so that the displacement thickness can be added to the original geometry after every viscous-inviscid interactive iteration.
- 3) The inviscid flow analysis, described in Section 4.2.1, is conducted. The freestream velocity and pressure coefficient distributions, and the location of the stagnation point are calculated.
- 4) The flow distance for each side of the airfoil, from the stagnation point to the trailing edge is calculated. Here, the non-displaced position of the node that is closest to the stagnation point is assumed to be the stagnation point, and the flow distance on each side is calculated using the original airfoil geometry.
- 5) The calculated flow distance is discretized for the viscous flow analysis, and the freestream velocity at every node is interpolated. Also, dU_e/ds for every node is calculated.
- 6) The viscous flow analysis is conducted as described in Section 4.2.2.

- 7) The aerodynamic coefficients are calculated and checked for convergence. If convergence is achieved, the calculation proceeds to the next flow condition (e.g., different airfoil, different angle of attack, or different Reynolds number). If convergence is not achieved, the panels representing the airfoil surface are displaced outwards using the under-relaxed displacement thickness distribution.
- 8) Steps 3-7 are repeated until convergence to the desired level is reached.

4.3 Calculation of the 3D Aerodynamics of Rotating Blades

When calculating the performance of blades with finite span, it is important to include the effects of induced velocity with the associated root and tip losses (Leishman, 2006). The present algorithm uses the blade-element-momentum theory with Prandtl root and tip losses to calculate the induced effects and the effective angle of attack of each blade element on a finite-span rotating blade. Using the effective angle of attack, one can assume each blade element behaves as a 2D airfoil under the same flow conditions. Stall delay corrections as provided by Corrigan & Schillings (1994) and Lindenburg (2004) are used to account for radial-flow effects. The calculation methods for wind turbines and propellers are presented separately, since the final equations are slightly different due to the direction of the induced velocity between the two applications being opposite compared to the undisturbed freestream direction.

4.3.1 Application of the Blade-Element-Momentum Theory for the 3D

Aerodynamic Analysis of Wind Turbine Blades

A cross-sectional view of a wind turbine blade is shown in Figure 4.6 with the aerodynamic force and moment coefficients and their corresponding directions as per the convention in the wind energy literature. The normal and tangential force coefficients, C_n and C_t , correspond to the force coefficients in the y' and $-x'$ directions, respectively, which are used in the 2D viscous-inviscid interaction method:

$$C_n = C_{y'} \quad (4.78)$$

$$C_t = -C_{x'} \quad (4.79)$$

Figure 4.6: Cross-sectional view of a wind turbine blade with aerodynamic force and moment coefficients. Reproduced from Simms et al. (2001).

4.3.1.1 Calculation of Induced Velocity for Wind Turbine Blades

The induced velocity on a wind turbine blade element is calculated using the induction factor of the blade-element-momentum theory from Eqn. (2.31), repeated here for convenience:

$$a(r, X_{\text{TSR}}) = -\sqrt{\left(\frac{\sigma X_{\text{TSR}} C_{l_\alpha}}{16} + \frac{1}{2}\right)^2 - \frac{\sigma X_{\text{TSR}} C_{l_\alpha} (X_{\text{TSR}} \theta_{\alpha_0} r + 1)}{8}} + \left(\frac{\sigma X_{\text{TSR}} C_{l_\alpha}}{16} + \frac{1}{2}\right)$$

where a is the induction factor, r is the nondimensional radial distance of the blade element from the rotation axis, X_{TSR} is the tip-speed ratio, σ is the local rotor solidity, C_{l_α} is $dC_l/d\alpha$ for the blade-element airfoil, and θ_{α_0} is the local pitch angle of the blade element with respect to the zero-lift angle of attack of the airfoil. To include the tip effects that alter the spanwise loading distribution on the blades, this equation is corrected using Prandtl's tip-loss function (Betz (1919)). This function approximates the effect of a finite number of blades and different blade twist distributions. As Leishman (2006) explains, Prandtl's function is expressed in terms of a correction factor, F , introduced into the thrust coefficient equation for an infinitesimally-thin rotor disc annulus (Eqn. (2.23)):

$$dC_T = 8Fa(1-a)r \, dr \quad (4.80)$$

in which

$$F = \left(\frac{2}{\pi}\right) \cos^{-1} \left(e^{-f}\right) \quad (4.81)$$

where the exponent f is given by:

$$f_{ip} = \frac{N_b}{2} \left(\frac{1-r}{r\phi}\right) = \frac{N_b}{2} \left(\frac{1-r}{1-a}\right) X_{\text{TSR}} \quad (4.82)$$

for tip losses, and

$$f_{root} = \frac{N_b}{2} \left(\frac{r-r_0}{1-a} \right) X_{TSR} \quad (4.83)$$

for root losses. In the above equations, dC_T is the thrust coefficient of the rotor disc annulus; N_b is the number of blades comprising the rotor; and $r_0 = y_0/R$ is the nondimensional radial root cut-out distance, which corresponds to the non-lift generating section of the blade close to the hub. With the Prandtl root- and tip-loss correction factor incorporated, the resulting induction factor equation is:

$$a(r, X_{TSR}, F) = -\sqrt{\left(\frac{\sigma X_{TSR} C_{l_\alpha}}{16F} + \frac{1}{2} \right)^2 - \frac{\sigma X_{TSR} C_{l_\alpha} (X_{TSR} \theta_{\alpha_0} r + 1)}{8F}} + \left(\frac{\sigma X_{TSR} C_{l_\alpha}}{16F} + \frac{1}{2} \right) \quad (4.84)$$

Eqn. (4.84) is only valid for $\alpha \leq 0.5$, i.e., when the following is true:

$$\left(\frac{\sigma X_{TSR} C_{l_\alpha}}{16F} - \frac{1}{2} \right)^2 \geq \frac{\sigma X_{TSR} C_{l_\alpha} (X_{TSR} \theta_{\alpha_0} r + 1)}{8F} \quad (4.85)$$

For $a > 0.5$, the corresponding equation including the correction factor, F , is:

$$a(r, X_{TSR}, F) = \sqrt{\left(\frac{\sigma X_{TSR} C_{l_\alpha}}{16F} - \frac{1}{2} \right)^2 - \left(\frac{1}{2} - \frac{\sigma X_{TSR} C_{l_\alpha} (X_{TSR} \theta_{\alpha_0} r + 1)}{8F} \right)} - \left(\frac{\sigma X_{TSR} C_{l_\alpha}}{16F} - \frac{1}{2} \right) \quad (4.86)$$

The induction factor for any given nondimensional spanwise location, r , is found in an iterative manner (Leishman, 2006), by using the above equations in the following way:

- 1) For $n=1$, where n is the number of iterations, $F^{n-1} = F^0 = 1$ is assumed and the induction factor a^1 is calculated using Eqn. (4.84) if the inequality in Eqn.

(4.85) is true, or Eqn. (4.86) if it is false. The calculation then advances to the next iteration, $n = 2$, and step 2.

- 2) The new correction factor, F^n , is calculated using a^{n-1} in Eqns. (4.81), and (4.82) if $r \geq (1+r_0)/2$, or (4.83) if $r < (1+r_0)/2$. The two inequality conditions correspond to the blade's lift-generating half-span close to the blade tip, and to the blade root, respectively.
- 3) The new induction factor a^n is calculated by using the new correction factor F^n in the appropriate equation (Eqn. (4.84) or (4.86)).
- 4) Steps 2 and 3 are repeated until convergence to the desired level is reached for the induction factor. In the present algorithm, convergence is assumed to be reached if $|a^n - a^{n-1}| < 0.001$.

Once the induction factor is calculated, the induced velocity, v_i , at the corresponding spanwise blade element is calculated by:

$$v_i = aV_\infty \quad (4.87)$$

where V_∞ is the undisturbed freestream velocity approaching the turbine i.e., the wind velocity.

4.3.1.2 Calculation of the Effective Angle of Attack

The effective angle of attack for a blade element is calculated by referring to Figure 2.3b. First, the inflow angle, ϕ , is calculated by using:

$$\phi = \tan^{-1} \left(\frac{V_\infty - v_i}{\Omega y} \right) \quad (4.88)$$

and the effective angle of attack is then calculated from

$$\alpha = \theta + \phi \quad (4.89)$$

where θ is the local pitch angle. The relative freestream velocity approaching the blade element is given by:

$$V_{\infty_r} = \sqrt{(V_{\infty} - v_i)^2 + (\Omega y)^2} \quad (4.90)$$

where Ω is the angular velocity of the rotor, and y is the radial distance of the blade element from the rotation axis. The freestream flow information, effective angle of attack, and the airfoil geometry for the spanwise blade element are then used in the 2D viscous-inviscid interaction method to calculate the aerodynamic coefficients of each blade element.

4.3.2 Application of the Blade-Element-Momentum Theory for the 3D Aerodynamic Analysis of Propeller Blades

4.3.2.1 Calculation of the Induced Velocity in Propeller Blades

Figure 4.7: Propeller blade element. Reproduced from Leishman (2006).

In Figure 4.7, a propeller blade element is shown with the front view of the blade being identical to the wind turbine model, shown in Figure 2.3a. Here, the total velocity in the axial direction is $V_\infty + v_i$.

The full derivation of the blade-element-momentum theory for propellers can be found in textbooks describing propeller or rotor aerodynamics such as Leishman (2006) or McCormick (1967), therefore it is not presented here. Instead, the final equations to calculate the induced effects including Prandtl's root- and tip-loss correction factor, F , are presented.

Instead of the induction factor, a , the inflow ratio, λ_i , is used for propellers:

$$\lambda_i = \frac{V_\infty + v_i}{\Omega R} \quad (4.91)$$

Here, V_∞ is the freestream velocity approaching the rotor, Ω is the angular velocity of the rotor, and R is the blade span. The equation to calculate the inflow ratio for a blade element, provided by Leishman (2006), is:

$$\lambda_i(r, \lambda_{i_\infty}, F) = \sqrt{\left(\frac{\sigma C_{l_\alpha}}{16F} - \frac{\lambda_{i_\infty}}{2}\right)^2 + \frac{\sigma C_{l_\alpha}}{8F} \theta_{\alpha_0} r} - \left(\frac{\sigma C_{l_\alpha}}{16F} - \frac{\lambda_{i_\infty}}{2}\right) \quad (4.92)$$

where r is the nondimensional radial distance of the blade element from the rotation axis, σ is the local rotor solidity, C_{l_α} is $dC_l/d\alpha$ for the blade-element airfoil, λ_{i_∞} is the freestream inflow ratio defined as:

$$\lambda_{i_\infty} = \frac{V_\infty}{\Omega R} \quad (4.93)$$

and θ_{α_0} is the local pitch angle of the blade element with respect to the zero-lift angle of attack of the airfoil. The Prandtl root- and tip-loss correction factor, F , is defined as:

$$F = \left(\frac{2}{\pi} \right) \cos^{-1} (e^{-f}) \quad (4.94)$$

where the exponent f is defined as:

$$f_{tip} = \frac{N_b}{2} \left(\frac{1-r}{r\phi} \right) \quad (4.95)$$

when calculating tip losses, and as

$$f_{root} = \frac{N_b}{2} \left(\frac{r}{(1-r)\phi} \right) \quad (4.96)$$

when calculating root losses. The inflow angle, ϕ , is calculated using:

$$\phi = \tan^{-1} \left(\frac{\lambda_i(r)}{r} \right) \quad (4.97)$$

The inflow ratio at any given nondimensional spanwise location, r , is found in an iterative manner as follows (Leishman, 2006):

- 1) If $n=1$, where n is the number of iterations, $F^{n-1} = F^0 = 1$ is assumed and the inflow ratio, λ_i^1 , is calculated using Eqn. (4.92). The calculation then advances to the next iteration, $n=2$, and step 2.
- 2) The new correction factor, F^n , is calculated from Eqn. (4.94) using Eqn. (4.95) if $r \geq (1+r_0)/2$, or Eqn. (4.96) if $r < (1+r_0)/2$, and using λ_i^{n-1} in Eqn. (4.97). The two inequality conditions correspond to the blade's lift-generating half-span close to the blade tip, and to the blade root, respectively.
- 3) The new inflow ratio, λ_i^n , is calculated by using the new correction factor, F^n , in Eqn. (4.92).

- 4) Steps 2 and 3 are repeated until convergence to the desired level is reached for the inflow ratio. In the present algorithm, convergence is assumed to be reached if $|\lambda_i^n - \lambda_i^{n-1}| < 0.0001$.

Once the inflow ratio is calculated, the induced velocity, v_i , at the corresponding spanwise location is calculated from the definition of the inflow ratio (Eqn. (4.91)):

$$v_i = \lambda_i \Omega R - V_\infty \quad (4.98)$$

4.3.2.2 Calculation of the Effective Angle of Attack

Once the induced velocity is calculated, inflow angle, ϕ , is calculated by using (Figure 4.7):

$$\phi = \tan^{-1} \left(\frac{V_\infty + v_i}{\Omega y} \right) \quad (4.99)$$

and the effective angle of attack is then calculated from:

$$\alpha = \theta - \phi \quad (4.100)$$

where θ is the local blade pitch angle. The undisturbed freestream velocity approaching the blade element is

$$V_{\infty_r} = \sqrt{(V_\infty + v_i)^2 + (\Omega y)^2} \quad (4.101)$$

As described for wind turbines, with this information, the 2D viscous-inviscid interaction method is used to calculate the aerodynamic coefficients of the corresponding spanwise propeller blade element.

4.3.3 Stall Delay Corrections

To account for the stall delay caused by radial flow (explained in Section 3.4.3), two models proposed by Corrigan & Schillings (1994) and Lindenburg (2004), described in Sections 3.4.3.1 and 3.4.3.2, respectively, are used. These models are used to modify the lift and normal force coefficients calculated by the 2D viscous-inviscid interaction method, respectively.

To calculate the lift coefficient using the Corrigan-Schillings model under rotating conditions at any blade element, Eqns. (3.2) and (3.3) are used to calculate the stall delay angle of attack, $\Delta\alpha$. The 2D lift coefficient for the blade element calculated by the viscous-inviscid interaction method is then modified using Eqn. (3.4) to calculate the lift coefficient under rotating conditions $C_{l, rot}$.

The Lindenburg stall delay model modifies the normal force coefficient, C_n (Eqn. (4.78)), to account for stall delay using Eqn. (3.5), repeated here for convenience:

$$C_{n, rot} = C_n + 1.6 \left(\frac{c}{y} \right) (1 - f_{sep}) \cos^2 \phi$$

where $C_{n, rot}$ is the normal force coefficient under rotating conditions; c is the chord length; y is the radial distance of the blade element from the rotation axis; ϕ is the inflow angle; and f_{sep} is the nondimensional chordwise location of flow separation, calculated as $f_{sep} = x'_{sep}/c$, in which x'_{sep} is the chordwise location of flow separation predicted by the viscous flow analysis.

4.4 Calculation of Rotor Performance

Referring to Figure 2.3b, the thrust force, torque, and power for the wind turbine rotor annulus created by a 360° sweep of a blade element around the rotor axis are calculated using the following equations once the lift and drag forces of the element are known:

$$dT = N_b (dL \cos \phi + dD \sin \phi) \quad (4.102)$$

$$dQ = N_b (dL \sin \phi - dD \cos \phi) y \quad (4.103)$$

$$dP = N_b (dL \sin \phi - dD \cos \phi) \Omega y \quad (4.104)$$

where dT , dQ , and dP are the thrust, torque, and power of the rotor annulus, respectively; N_b is the number of blades; dL and dD are the lift and drag forces produced by the blade element, respectively; Ω is the angular velocity of the blade; and y is the radial distance of the blade element from the rotation axis.

For propellers, referring to Figure 4.7, the corresponding equations are:

$$dT = N_b (dL \cos \phi - dD \sin \phi) \quad (4.105)$$

$$dQ = N_b (dL \sin \phi + dD \cos \phi) y \quad (4.106)$$

$$dP = N_b (dL \sin \phi + dD \cos \phi) \Omega y \quad (4.107)$$

Once the performance parameters of all rotor annuli are calculated, they are numerically integrated along the radial distance from the root cut-out, y_0 , to the blade tip:

$$T = \int_{y_0}^R dT \, dy \quad (4.108)$$

$$Q = \int_{y_0}^R dQ dy \quad (4.109)$$

$$P = \int_{y_0}^R dP dy \quad (4.110)$$

in which R is the blade span. The total thrust and power for a wind turbine are expressed in coefficient form, C_T and C_P , by using Eqns. (2.10) and (2.12), respectively. The torque coefficient is expressed as:

$$C_Q = \frac{Q}{\frac{1}{2} \rho A V_\infty^2 R} \quad (4.111)$$

in which ρ is the density and A is the rotor disc area. For propellers, the corresponding equations for C_T , C_P , and C_Q are:

$$C_T = \frac{T}{\rho n^2 (2R)^4} \quad (4.112)$$

$$C_P = \frac{P}{\rho n^3 (2R)^5} \quad (4.113)$$

$$C_Q = \frac{Q}{\rho n^2 (2R)^5} \quad (4.114)$$

where n is the number of rotations per second. These performance measures are used to evaluate the overall effects of design changes, hence they are performed as the final step in the blade performance calculation.

In this chapter, the algorithm for analyzing rotating-blade aerodynamics based on a 2D weak viscous-inviscid interaction method and the blade-element-momentum theory has been presented. In the following chapter, the prediction capability of the present algorithm is discussed.

Chapter 5: Verification and Validation

To assess the capabilities and limitations of the present algorithm, the predictions are compared to numerical and experimental data from published literature. Sections 5.1 and 5.2 verify and validate the inviscid and viscous flow analysis algorithms, respectively. In Section 5.3, the predicted 2D aerodynamic characteristics of the S809 and NACA 0012 airfoils are compared to experimental data. Finally in Section 5.4, the predicted aerodynamic performance of a rotating wind turbine blade is compared to experimental data from the Unsteady Aerodynamics Experiment (UAE) Phase VI (Hand et al., 2001), conducted by the National Renewable Energy Laboratory (NREL).

No uncertainty information is available for the experimental data presented in this chapter. The experimental data points were obtained by digitizing published figures, and the inherent error of this process is expected to result in an uncertainty of about $\pm 5\%$, which likely exceeds the uncertainty margins of the experimental data.

5.1 Inviscid Flow Analysis Algorithm

To verify and validate the inviscid flow analysis algorithm, the pressure coefficient distribution predicted by the present source-vortex panel method on the top (or bottom) half of a 2D cylinder in cross-flow is compared to the analytical solution. The comparison is shown in Figure 5.1, where ϕ is the angular position around the cylinder with 0° corresponding to the angular location of the rear stagnation point. In the figure, two cases with an angle difference between adjacent panels of 10° and 20° , respectively, are presented. These correspond to 36 and 18 panels along the circumference of the cylinder, with corresponding maximum error in the local C_p predictions as percent of

$C_{p_{max}} = 1$ is 3% and 10%. As discussed in Section 4.2.1.5, a maximum difference of 10° between adjacent panels is recommended. Additional verification and validation of the inviscid flow analysis algorithm is presented in Sections 5.3.1.1 and 5.3.2.1 by comparing the predictions against experimental data for the S809 and NACA 0012 airfoils.

Figure 5.1: C_p distribution on one half of a 2D cylinder in cross-flow.

5.2 Viscous flow Analysis Algorithm

5.2.1 Laminar Momentum-Integral Predictions

A zero-pressure-gradient laminar boundary layer calculated using Thwaites' laminar momentum-integral closure method is compared to the similarity solution of Blasius (1907) in Figure 5.2a, where s/L indicates streamwise distance nondimensionalized by the surface length. The predictions agree to within less than 1% of the local values in θ , δ^* , and C_f . A similar comparison is shown in Figure 5.2b against the similarity solution provided by White (2006) for a Falkner & Skan (1931)

self-similar flow past a wedge with an angle of 27° . In this case, the momentum and displacement thicknesses predicted by the two methods agree to within 3% of the local values. The discrepancy is within the 5% accuracy range suggested by White (2006) for Thwaites' (1949) method. These results verify the correct implementation of Thwaites' momentum-integral closure method and validate the method for accurately predicting laminar boundary-layer development.

Figure 5.2: Comparison of the present laminar momentum-integral predictions against (a) Blasius' similarity solution and (b) the similarity solution for a Falkner-Skan self-similar flow past a 27° wedge.

5.2.2 Turbulent Momentum-Integral Predictions

Figure 5.3 shows that the skin-friction coefficient, C_f , for a turbulent zero-pressure-gradient boundary layer predicted by the present turbulent momentum-integral closure algorithm agrees with the predictions of Prandtl's (1921) power law approximation to within 8% of the local value, excluding the peak values upstream of $s/L = 0.04$. In this case, θ and δ^* agree to within less than 3% and 8% of the local values, respectively. The discrepancies are due to the streamwise distance required for the

momentum-integral predictions to converge to the equilibrium zero-pressure-gradient values.

Figure 5.3: Comparison of predicted C_f against Prandtl's power-law approximation for a zero-pressure-gradient turbulent boundary layer.

Figure 5.4: Prediction of C_f in a zero-pressure-gradient turbulent boundary layer.

Figure 5.5: Prediction of Re_θ and Re_{δ^*} in a zero-pressure-gradient turbulent boundary layer.

Favourable agreements are also noted in Figure 5.4, where the prediction of C_f from the present algorithm is shown to be in agreement to within 5%, 12%, and 7% of the DNS results of Spalart (1988), the experimental data of Coles (1964), and other experimental results (Purtell et al. (1981), Murlis et al. (1982), and Erm et al. (1985)) presented in Spalart's paper, respectively. The discrepancy between the predictions and Coles' (1964) results is likely due to the Coles' curve-fit approach to approximate different sets of experimental data from a number of studies, which, in Coles' paper is shown to overshoot experimental C_f values at small Re_θ , and undershoot at high Re_θ . Since the present turbulent momentum-integral closure is an approximate method, some discrepancy is expected when comparing against experimental studies. As observed in Figure 5.5, the streamwise variation of displacement and momentum thicknesses

predicted by the present algorithm agree favourably with those of Spalart (1988) obtained through DNS.

5.2.3 Boundary Layers with Attached-Flow Transition

The boundary-layer development predicted by the present algorithm for transitioning boundary layers under adverse (APG), zero (ZPG), and favourable (FPG) pressure gradients are compared to the experimental results provided by Abu-Ghannam & Shaw (1980) in Figures 5.6, 5.7, and 5.8, which plot Re_θ , C_f , and H , respectively. The approximate experimental locations of transition onset and completion are also shown. The freestream turbulence intensities for the APG, ZPG, and FPG cases are 1.20%, 1.25%, and 1.55% , respectively.

Figure 5.6: Prediction of Re_θ for boundary layers with attached-flow transition.

Figure 5.6 shows reasonable agreement with the experimental data for the fully-laminar and transitional regions. The under-prediction in the turbulent region is noted to be most severe for the APG case, with the FPG case having the least error, which is not surprising since it is generally more difficult to predict boundary-layer parameters for adverse pressure gradients. However, the good agreement in Figure 5.5 for the comparison against Spalart's DNS for a ZPG turbulent flow suggests that the discrepancies noted in the turbulent region in Figure 5.6 are likely due to the prediction accuracy in the transition region, for the predicted flow conditions at the end of the transition region form the initial conditions for the turbulent region.

Figure 5.7: Prediction of C_f for boundary layers with attached-flow transition.

Figure 5.7 shows that the predicted skin-friction coefficients are in good agreement with the experimental data in the laminar region. The increase in skin friction with transition inception and the decrease in skin friction with streamwise distance upon transition completion are generally predicted with sufficient accuracy. The late transition onset predicted by the present algorithm in the FPG case is noted to result from Abu-

Ghannam & Shaw's (1980) model for predicting the transition inception location (Eqn. (4.48)), which suggests further improvements to the model are possible.

Figure 5.8: Prediction of H for boundary layers with attached-flow transition.

Figure 5.8 shows that the shape factor is under-predicted in the laminar and turbulent regions for all cases, while a better agreement is present for the transitional region. However, the predicted shape factor values are in agreement with the general trend; a higher shape factor indicates a more-adverse pressure gradient, and vice versa. The experimental shape factor in the fully-turbulent region for the FPG case is noted to be almost equal to the APG case, which disagrees with the general trend. The discrepancies in Figure 5.8 are therefore attributed to limitations of the present laminar and turbulent momentum-integral closure methods, and the limitations in matching the conditions of the experiments within the simulations.

Additional verification of the transition model proposed by Roberts & Yaras (2005b) was performed during the implementation of the model into the present algorithm, and since the transition model was thoroughly validated during its development, additional validation was not deemed necessary. The comparisons presented in this section demonstrate that the momentum-integral method is a suitable choice for the viscous flow analysis, providing sufficiently accurate predictions for the boundary-layer parameters that are most relevant to the aerodynamic performance of an airfoil.

5.3 Weak Viscous-Inviscid Interaction Algorithm for Predicting 2D Airfoil

Performance

In this section, the 2D aerodynamic performance predicted using the present viscous-inviscid interaction algorithm for the S809 and NACA 0012 airfoils are compared to published experimental data. The rationale for selecting these airfoils is provided in Section 4.2.1.5. The test cases are summarized in Table 5.1.

Table 5.1: Test cases for the verification and validation of the 2D viscous-inviscid interaction algorithm.

Case ID	Airfoil	Re_c	Angles of attack tested	Parameters studied
S1	S809	1×10^6	$-2^\circ \leq \alpha \leq 20^\circ$	C_l, C_d, C_m
S2	S809	2×10^6	$-2^\circ \leq \alpha \leq 20^\circ$	C_p, C_l, C_d, C_m
N1	NACA 0012	3×10^6	$-2^\circ \leq \alpha \leq 20^\circ$	C_p, C_l, C_d, C_m

5.3.1 S809 Airfoil

5.3.1.1 Airfoil Surface Pressure Distributions

The predicted pressure distributions for the S809 airfoil for Case S2 at 5.1° , 10.2° , and 20.2° angle of attack, which correspond to unstalled, lightly-stalled, and heavily-stalled conditions, respectively, are compared to the experimental results of Somers (1997) in Figures 5.9, 5.10, and 5.11, respectively. The pressure distribution predicted from an inviscid-flow-only solution is also shown for comparison. The predicted transition mode and nondimensional chordwise (x'/c) separation, transition inception, and transition completion locations for the two airfoil surfaces at 5.1° , 10.2° , and 20.2° angle of attack in Case S2 are presented in Tables 5.2, 5.3, and 5.4, respectively, along with the transition mode and turbulent reattachment location of the separation bubbles observed by Somers (1997).

Figure 5.9: C_p distribution for Case S2 at $\alpha = 5.1^\circ$.

Table 5.2: Transition mode and nondimensional chordwise separation, transition inception, and transition completion locations for Case S2 at $\alpha = 5.1^\circ$.

Airfoil Surface	Transition mode		Laminar sep. (Pred.)	Transition inception (Pred.)	Transition completion		Separation (Pred.)
	Pred.	Exp.			Pred.	Exp.	
Suction	Attached	Sep. bubble	-	0.280	0.531	0.509	0.949
Pressure	Sep. bubble	Sep. bubble	0.461	0.462	0.476	0.515	-

The C_p distribution for $\alpha = 5.1^\circ$ (Figure 5.9) shows good agreement with experimental results, except for some discrepancy around mid-chord, where Somers indicates separation bubbles cause the sharp increase in C_p on both sides of the airfoil. As presented in Table 5.2, instead of the separation bubble-transition observed by Somers, the present algorithm predicts attached-flow transition on the suction side, which results in a more gradual C_p trend. Although the separation-bubble transition mode on the pressure side is predicted correctly, the predicted C_p distribution does not sharply increase as observed by Somers, which suggests the present method for estimating the displacement thickness within separation bubbles may be improved. As expected for unstalled angles of attack, the effect of boundary-layer development on the inviscid stream surrounding the boundary layer is minimal, as illustrated by the favourable agreements between the inviscid and viscous-inviscid interactive solutions.

Figure 5.10: C_p distribution for Case S2 at $\alpha = 10.2^\circ$.

Table 5.3: Transition mode and nondimensional chordwise separation, transition inception, and transition completion locations for Case S2 at $\alpha = 10.2^\circ$.

Airfoil Surface	Transition mode		Laminar sep. (Pred.)	Transition inception (Pred.)	Transition completion		Separation (Pred.)
	Pred.	Exp.			Pred.	Exp.	
Suction	Sep. bubble	Sep. bubble	0.020	0.033	0.043	0.025	0.773
Pressure	Sep. bubble	Sep. bubble	0.490	0.495	0.514	0.534	-

The pressure distribution for Case S2 at $\alpha = 10.2^\circ$ (Figure 5.10) shows reasonable agreement with experimental results for the suction side, and good agreement for the pressure side. The separation-bubble transition on the suction side is predicted with good accuracy (Table 5.3); however, the trailing-edge separation location is predicted further downstream than the location suggested by the experimental C_p distribution in Figure 5.10, close to $x'/c = 0.55$. Although the predictions of the viscous-inviscid

interaction algorithm with the stalled-aerodynamics prediction method are more accurate compared to the inviscid flow solution, the accuracy of the predictions decrease as the angle of attack increases, as explained in the following sections.

The limitations of the stalled-aerodynamics prediction method are clearly visible in Figure 5.11, which shows the pressure distribution for $\alpha = 20.2^\circ$. Although the pressure side shows excellent agreement, the agreement on the suction side is poor. It is shown in the following sections that this disagreement in the suction-side pressure distribution causes poor drag and pitching-moment predictions at heavily-stalled angles of attack; the drag coefficient is under-predicted and the pitching moment is over-predicted. This suggests that the inviscid pressure distribution around the assumed displacement body is not equal to the actual pressure distribution on the airfoil surface, and thus the assumption of a uniform pressure distribution between the separated shear layer and the airfoil surface is not valid in heavily-stalled conditions.

Figure 5.11: C_p distribution for Case S2 at $\alpha = 20.2^\circ$.

Table 5.4: Transition mode and nondimensional chordwise separation, transition inception, and transition completion locations for Case S2 at $\alpha = 20.2^\circ$.

Airfoil Surface	Transition mode		Laminar sep. (Pred.)	Transition inception (Pred.)	Transition completion		Separation (Pred.)
	Pred.	Exp.			Pred.	Exp.	
Suction	Sep. bubble	N/A	0.029	0.030	0.064	N/A	0.598
Pressure	Sep. bubble	N/A	0.550	0.559	0.592	N/A	-

5.3.1.2 Lift Coefficient

The predicted lift coefficients for the S809 airfoil in Cases S1 and S2 are compared to the experimental data of Ramsay et al. (1995) and Somers (1997) in Figure 5.12. Good agreement with Somers' data is present for the unstalled region ($\alpha < 7^\circ$), where over-prediction remains less than 10% of $C_{l_{max}}$. Between $7^\circ < \alpha < 10^\circ$, a change is visible in the experimental lift-curve slope indicating flow separation at the trailing edge, which prompts the use of the stalled-aerodynamics prediction method for angles of attack beyond $\alpha \geq 7^\circ$. For the stalled angle of attack range of $10^\circ \leq \alpha \leq 20^\circ$, reasonable agreement exists between the predictions of the present algorithm and experimental data, which demonstrates the level of accuracy attainable by the stalled-aerodynamics prediction method. It can be seen that the predicted lift coefficient beyond stall increases with Reynolds number, which is characteristic of most airfoils (Abbott & von Doenhoff, 1959), and can also be seen in Somers' data. It is recommended that the present algorithm not be used for angles of attack greater than about 20° , since the predictions and experiments do not agree well beyond this angle of attack, and the algorithm is not optimized for use beyond $\alpha = 20^\circ \sim 25^\circ$.

Figure 5.12: C_l versus α for the S809 airfoil for Cases S1 and S2.

In Figure 5.13, the lift coefficient distribution obtained for Case S2 using the conventional approach of assuming a uniform pressure distribution downstream of the separation point are compared with the experimental results of Somers (1997). The under-prediction of the lift coefficient prevents the usage of the uniform- C_p approach for modelling separated flow and necessitates using the stalled-aerodynamics prediction method presented in Section 4.2.2.7.

Figure 5.13: C_l vs. α for Case S2, using uniform- C_p approach downstream of flow separation.

5.3.1.3 Drag Coefficient

The drag coefficients predicted for the S809 airfoil are compared to the experimental results of Ramsay et al. (1995) and Somers (1997) in Figure 5.14 for Cases S1 and S2. The experimental drag coefficients were obtained by means of wake traversing to capture the effects of both pressure and skin-friction drag. The predictions in the unstalled region show good agreement with experimental data. The effect of Reynolds number on the drag coefficients is also captured, with the C_d predictions for Case S1 being higher than Case S2. The agreement worsens beyond stall due to the difficulty of modelling separated flow. This result leads to the conclusion that the aerodynamic coefficients predicted under stalled conditions are only usable for qualitative analysis for about $3^\circ \sim 5^\circ$ angle of attack beyond stall.

Figure 5.14: C_d vs. α for Cases S1 and S2.

5.3.1.4 Pitching-Moment Coefficient

Figure 5.15 compares the predicted pitching-moment coefficient for Cases S1 and S2 to the experimental results of Ramsay et al. (1995) and Somers (1997). The results show a reasonable agreement up to $\alpha = 7^\circ$, beyond which stall occurs and the pitching-moment coefficients are over-predicted. The over-prediction under stalled conditions is due to the C_p distribution on the suction side being under-predicted near the leading edge and over-predicted near the trailing edge, resulting in a higher pitching moment. It is noted that variations in flow Reynolds number do not have a significant effect on the pitching-moment coefficients, which can be seen from both the experimental and predicted data.

Figure 5.15: C_m vs. α for Cases S1 and S2.

5.3.2 NACA 0012 Airfoil

5.3.2.1 Airfoil Surface Pressure Distributions

The predicted pressure distributions for the NACA 0012 airfoil for Case N1 at 9.9° and 14.9° angle of attack are compared to the experimental results of Harris (1981) in Figures 5.16 and 5.17, respectively. The uncertainty of this particular set of experimental data is estimated to be about ± 0.1 for C_p . The C_p distributions at both angles of attack show good agreement with experimental data, and are within the estimated uncertainty range of the experimental data. Since both distributions are for an unstalled angle of attack, the predictions of the viscous-inviscid interaction algorithm and the inviscid-only solution are almost identical.

Figure 5.16: C_p distribution for Case N1 at $\alpha = 9.9^\circ$.

Figure 5.17: C_p distribution for Case N1 at $\alpha = 14.9^\circ$.

5.3.2.2 Lift Coefficient

Comparison of the predicted lift coefficient with experimental results provided by Abbott & von Doenhoff (1959) is presented in Figure 5.18. The stalled-aerodynamics prediction method is used for angles of attack beyond $\alpha = 15^\circ$. The results show good agreement for $\alpha \leq 15^\circ$, and reasonable agreement for greater angles of attack. The slight over-prediction around $\alpha = 13^\circ - 14^\circ$ is attributed to the C_p distribution on the suction side leading edge being under-predicted, as shown in Figure 5.17.

Figure 5.18: C_l vs. α for the NACA 0012 airfoil at $Re_c = 3 \times 10^6$.

5.3.2.3 Drag Coefficient

Figure 5.19 presents the drag coefficient up to $\alpha = 12^\circ$. The excellent agreement with the experimental results demonstrate the accuracy of drag coefficient predictions. Although the lack of published experimental data in stalled conditions prevents validation at high angles of attack, it is expected that the drag coefficient for angles of attack greater

than the stall angle ($\alpha_{C_{l_{max}}} = 15^\circ$) will be under-predicted, similar to the trend observed for the S809 airfoil. The discontinuity in the slope of the predicted drag curve is due to variations in the predicted drag coefficients within the uncertainty range of the present algorithm.

Figure 5.19: C_d vs. α for the NACA 0012 airfoil at $Re_c = 3 \times 10^6$.

5.3.2.4 Pitching-Moment Coefficient

Figure 5.20 presents the pitching-moment coefficient up to $\alpha = 18^\circ$. The predictions agree well with experimental data up to the stall angle, beyond which the pitching moment is over-predicted. The same trend is observed for the S809 airfoil, and the reason for this is likely the pressure distributions predicted under stalled conditions not being accurate.

Figure 5.20: C_m vs. α for the NACA 0012 airfoil at $Re_c = 3 \times 10^6$.

5.4 Aerodynamic Loads and Performance of a Rotating Wind Turbine Blade

To evaluate the accuracy of the aerodynamic load predictions for a rotating wind turbine blade, a comparison was made for the normal, tangential, and pitching-moment coefficients measured in the UAE - Phase VI (Hand et al., 2001), conducted by the NREL. Since this experiment was performed in a wind tunnel, the unsteady effects in the atmosphere such as freestream turbulence and gusts were not present, allowing for the results of the experiment to be directly compared to the predictions of the present algorithm.

The rotor and operating condition data for the NREL test turbine is provided by Fletcher et al. (2009), and the geometry of the turbine blade is provided by Hand et al. (2001), which are summarized in Tables 5.5 and 5.6, respectively.

Table 5.5: Rotor data and operating conditions for the NREL test turbine.

Number of blades	2
Rotor radius	5.029 m
Root-cutout radius	1.257 m
Airfoil	NREL S809
Rotational speed	72 RPM (constant for all wind speeds)
Blade tip pitch	3° towards feather

Table 5.6: Blade geometry of the NREL test turbine (lift-generating sections only).

Radial distance, y (m)	$r = y/R$	Chord length, c (m)	Twist¹ (°)	Maximum Thickness
1.343	0.267	0.728	18.074	20.95% c
1.51	0.300	0.711	14.292	20.95% c
1.648	0.328	0.697	11.909	20.95% c
1.952	0.388	0.666	7.979	20.95% c
2.257	0.449	0.636	5.308	20.95% c
2.343	0.466	0.627	4.715	20.95% c
2.562	0.509	0.605	3.425	20.95% c
2.867	0.570	0.574	2.083	20.95% c
3.172	0.631	0.543	1.15	20.95% c
3.185	0.633	0.542	1.115	20.95% c
3.476	0.691	0.512	0.494	20.95% c
3.781	0.752	0.482	-0.015	20.95% c
4.023	0.800	0.457	-0.381	20.95% c
4.086	0.812	0.451	-0.475	20.95% c
4.391	0.873	0.42	-0.92	20.95% c
4.696	0.934	0.389	-1.352	20.95% c
4.78	0.950	0.381	-1.469	20.95% c
5	0.994	0.358	-1.775	20.95% c

The data for the normal and tangential force coefficients at five spanwise locations are provided by Fletcher et al. (2009), and the pitching-moment coefficients at

¹ Angle convention in wind turbine literature is positive toward feather.

three spanwise locations are provided by Simms et al. (2001) for an unyawed configuration at wind speeds of 7 m/s, 10 m/s, and 25 m/s, which correspond to unstalled, lightly-stalled (less than 5° over $\alpha_{C_{l_{max}}}$), and heavily-stalled conditions (more than 5° over $\alpha_{C_{l_{max}}}$), respectively.

To predict the 3D aerodynamic performance of the blade, a database containing 2D aerodynamic performance parameters (aerodynamic coefficients, transition inception, transition completion, and boundary-layer separation locations) for the airfoils used along the span of the rotor blade is created, with the performance of each airfoil section established for the range of angles of attack estimated by the blade-element momentum theory of 1° increments, and flow Reynolds number of 250,000 increments. The Reynolds number increment is deemed sufficient because the change in aerodynamic coefficients between two increments is not more than 3% for a given angle of attack. To compute the 3D performance of the rotor blade, the algorithm makes use of the parameters stored in the 2D airfoil performance database by linearly interpolating the aerodynamic parameters using the angle of attack and Reynolds number of each blade element.

5.4.1 Normal and Tangential Forces

The normal and tangential force coefficients, C_n and C_t , for a wind speed of $V_\infty = 7$ m/s are presented in Figure 5.21. At this configuration, the angle of attack predicted by the blade-element-momentum theory for different blade elements ranges

from 1.7° to 7.5° , whereas the Reynolds number ranges from $Re_c = 5.5 \times 10^5$ to $Re_c = 9.4 \times 10^5$ over the blade span.

Figure 5.21: Spanwise C_n and C_t distributions for the NREL UAE Phase VI blade at $V_\infty = 7$ m/s.

The predictions with no stall delay correction and the predictions with the Lindenburg stall delay correction show reasonable agreement with experimental data, with some over-prediction of the normal force coefficient from mid-span to the blade tip. In comparison to the Lindenburg stall delay model, the Corrigan-Schillings model over-predicts the normal force coefficient, which is not surprising, since the Lindenburg model is specifically calibrated to match the results of the NREL Phase VI experiment. Furthermore, the Lindenburg model directly uses the boundary-layer separation location predicted by the 2D viscous-inviscid interaction algorithm, whereas the Corrigan-Schillings model predicts the separation location using an empirical correlation (Eqn.

(3.2)). Therefore it is expected that the Lindenburg model will perform better for an application that neither of the stall delay models are specifically calibrated for.

Figure 5.22: Spanwise C_n and C_t distributions for the NREL UAE Phase VI blade at $V_\infty = 10$ m/s.

In Figure 5.22, the normal and tangential force coefficients for a wind speed of $V_\infty = 10$ m/s are presented. In this configuration, the predicted angle of attack of the blade elements ranges from 3.7° to 14.2° , and the Reynolds number ranges from $Re_c = 5.7 \times 10^5$ to $Re_c = 9.6 \times 10^5$. The results show reasonable agreement for the predicted normal force coefficients using stall delay corrections, and good agreement for the tangential force coefficient, neglecting the experimental data for the tangential force at $y/R = 0.47$. In this lightly-stalled condition, the benefit of stall delay corrections become clear, since regions of separated flow promote spanwise motion. The difference between the predictions of the two stall delay models is also less compared to Figure 5.21, which indicates that the Corrigan-Schillings model does a reasonably good job

under stalled conditions, even though it has not been precisely calibrated for this data set. The agreement in Figure 5.22 confirms the usability of the present algorithm for the performance and load prediction of rotating blades under lightly-stalled conditions.

Figure 5.23: Spanwise C_n and C_l distributions for the NREL UAE Phase VI blade at $V_\infty = 25$ m/s.

In Figure 5.23, the normal and tangential force coefficients for a wind speed of $V_\infty = 25$ m/s are presented. Under this heavily-stalled condition, the angle of attack of the blade elements ranges from 16.2° to 40.5° , and the Reynolds number ranges from $Re_c = 8.2 \times 10^5$ to $Re_c = 1.2 \times 10^6$. Poor agreement between the predictions and the experimental data is expected due to the viscous-inviscid interaction method for the 2D aerodynamics having very limited accuracy under stalled conditions, and the blade-element-momentum theory being based on the assumption that each spanwise blade element can be treated independently without any spanwise flow effects, which become significant under stalled conditions. The poor predictions for C_n and C_l are mainly due

to the under-prediction of the drag coefficient at stalled angles of attack causing the normal force coefficient to be under-predicted, and the tangential force coefficient to be over-predicted.

5.4.2 Pitching Moment

Figure 5.24: Spanwise C_m distribution for the NREL UAE Phase VI blade at $V_\infty = 7$, $V_\infty = 10$, and $V_\infty = 25$ m/s.

The predicted and experimental pitching-moment coefficients for wind speeds of 7, 10, and 25 m/s are compared in Figure 5.24. Good agreement is present for $V_\infty = 7$ m/s, with the slight discrepancies at the blade root and tip likely being caused by the effects of root and tip loss on the pitching moment not being accurately predicted by Prandtl's correction factor. The discrepancy at higher wind speeds is due to poor pitching-moment predictions by the viscous-inviscid interaction algorithm under stalled conditions, and the lack of empirical models that account for the change in the pitching

moment due to spanwise flow. Even if 2D pitching-moment data for the airfoil were available for stalled conditions, the absence of such empirical models makes it necessary to use methods that can simulate the 3D aerodynamics in order to accurately predict the pitching moment for rotating blades under such conditions.

The results presented in this chapter show that the present viscous-inviscid interaction algorithm can be used to predict the aerodynamic performance of 2D airfoil sections and rotating blades under incompressible, unstalled, and unyawed (for rotors) conditions, with sufficient accuracy for design optimization. The 2D aerodynamic coefficient (C_l , C_d , and C_m) predictions under these conditions are expected to be within 10% of the actual values. For stalled conditions, the predictions have limited accuracy. Even with a more advanced proposed approach to account for the effects of stall on airfoil performance, quantitatively accurate predictions are limited to only a few degrees of angle of attack beyond stall.

5.5 Computation Time

Since one of the goals of the present study is to ensure the developed algorithm is computationally efficient and suitable for quickly evaluating the effect of design changes to a 2D or 3D lift-generating body, the computation times were benchmarked using an Intel® Core™ i5-750 desktop processor running at a clock speed of 2.66 GHz. The CPU time required to analyze a wind turbine blade with a single nondimensional airfoil geometry along its span is expected to be within 10 minutes for unstalled operating conditions, and 15 minutes for lightly-stalled conditions, including the creation of the database for the 2D airfoil performance using the viscous-inviscid interaction algorithm.

The computation time increases for stalled conditions because of the increased angle of attack range over the blade span, and the requirement for a greater number of iterations for the 2D viscous-inviscid interaction algorithm to converge. Subsequent computations should require much less time under similar freestream conditions (turbulence intensity, integral length scale of turbulence, etc.), provided that the airfoil geometry does not change, as the computed aerodynamic characteristics are saved in a lookup table and can be accessed by the software when needed. For a 2D airfoil section represented by 120 and 220 panels, each viscous-inviscid interactive calculation iteration is estimated to take about 0.5 to 1 second of CPU time, respectively, so that computing the 2D airfoil aerodynamic performance at a given angle of attack and Reynolds number requires a maximum of 25 to 50 seconds of CPU time, respectively. An increase in computation speed of about 50% may be achieved by compiling the software and running it as a stand-alone executable instead of running it in a MATLAB[®] environment.

In this chapter, the prediction capability and the prediction accuracy of the present algorithm have been discussed by comparing the predictions with published experimental and numerical data, and information on the computational power and computation time requirements has been provided.

Chapter 6: Conclusions and Future Work

6.1 Conclusions

A comprehensive algorithm for calculating the aerodynamic performance of rotating blades under steady and incompressible flow conditions has been developed. State-of-the-art methods for modeling boundary-layer transition and stall delay have been combined with well-established aerodynamic analysis methods to provide a unique algorithm that strikes a good balance between accuracy, flow information, and computational efficiency. It is possible to integrate the present algorithm with a structural analysis algorithm to perform aeroelastic simulations of lift-generating bodies, which ensures that the present algorithm will retain its value as a useful design tool in the future among other CFD methods (RANS, LES, DNS) which do not offer the same capability. The weak viscous-inviscid interaction scheme of the algorithm allows the viscous effects to be accounted for, so that the skin-friction and pressure drag can be predicted along with the lift and pitching moment. The algorithm is shown to provide predictions that are sufficiently accurate for design optimization under unstalled operating conditions.

The present study also proposes and tests a new approach to modelling regions of separated flow in order to predict the aerodynamics of stalled airfoils. It is shown that under stalled operating conditions, the proposed stalled-aerodynamics prediction method may be used for a qualitative analysis of the aerodynamic performance of the blade; however, its accuracy is limited to about $3^\circ \sim 5^\circ$ angle of attack beyond stall. Based on the predictions of the present algorithm, it can be concluded that the complex and unsteady nature of stalled operating conditions requires the use of more detailed methods such as RANS simulations.

6.2 Future Work

In order to increase the accuracy of the 2D aerodynamic predictions under stalled conditions, the weak viscous-inviscid interaction scheme may be converted to a strong interaction scheme by either simultaneously solving the viscous and inviscid flow equations or using the quasi-simultaneous solution method of Veldman (1981). However, with the 3D effects that are present under rotating and stalled conditions, the outcome may not be worth the time and effort invested in it, since the computationally-efficient methods available today for calculating the 3D aerodynamics, such as the blade-element-momentum theory, Prandtl's tip loss function, and empirical stall delay corrections, would still significantly limit the accuracy of the predictions under such conditions.

To make the stalled-aerodynamics prediction method applicable to all airfoils without the need to modify the coefficients of the close-to-linear correlation for m for each airfoil, a stall detection method (possibly based on the chordwise location of flow separation) can be developed to identify the angle of attack at which stall occurs, and a global equation to obtain the coefficients of the correlation can be identified by analyzing a wide range of airfoils with different thickness and camber properties. Although this process is straightforward, the limited usability of the stalled-aerodynamics prediction method prompts the improvement of the present algorithm in other directions.

As an example of such an improvement, instead of the 2D panel method used in the present algorithm, a 3D panel method may be used to model the full blade and its wake under rotating conditions to account for the 3D lift- and wake-induced flow effects, along with the present viscous flow analysis algorithm in a weak viscous-inviscid interaction scheme.

Appendices

Appendix A: Flowchart of the Present Aerodynamic Analysis Algorithm

Appendix B: Coordinates for the S809 and NACA 0012 Airfoils

B.1 S809 Airfoil

Figure B.1: S809 airfoil geometry.

Table B.1: Set of S809 airfoil coordinates provided by Somers (1997) with extra coordinates calculated in the present study using a piecewise cubic Hermite interpolating polynomial shaded in gray.

Upper Surface		Lower Surface	
x'/c	y'/c	x'/c	y'/c
0.00000	0.00000	0.00000	0.00000
0.00019	0.00158	0.00070	-0.00247
0.00037	0.00275	0.00140	-0.00498
0.00115	0.00862	0.00527	-0.01500
0.00575	0.01166	0.00933	-0.01272
0.01035	0.01708	0.01627	-0.01725
0.01626	0.02133	0.02321	-0.02162
0.02392	0.02673	0.03272	-0.02572

0.03158	0.03136	0.04223	-0.03144
0.04159	0.04149	0.05401	-0.04165
0.05147	0.04143	0.06579	-0.04199
0.06358	0.05145	0.07952	-0.05301
0.07568	0.05132	0.09325	-0.05301
0.08972	0.05630	0.10861	-0.06408
0.10390	0.06082	0.12397	-0.06408
0.11988	0.06972	0.14025	-0.07467
0.13580	0.06972	0.15752	-0.07467
0.15307	0.07786	0.17357	-0.08447
0.17103	0.07786	0.19362	-0.08447
0.19012	0.08505	0.21269	-0.09326
0.20920	0.08505	0.23175	-0.09326
0.22958	0.09113	0.25152	-0.10060
0.24987	0.09113	0.27129	-0.10060
0.27123	0.09594	0.29159	-0.10589
0.29259	0.09594	0.31188	-0.10589
0.31474	0.09933	0.33258	-0.10866
0.33689	0.09933	0.35328	-0.10866
0.35956	0.10050	0.37435	-0.10866
0.38223	0.10109	0.39541	-0.10842
0.40516	0.10107	0.41687	-0.10717
0.42809	0.10101	0.43832	-0.10484
0.45097	0.10015	0.46033	-0.10165
0.47384	0.09843	0.48234	-0.09756
0.49693	0.09583	0.50536	-0.09216
0.52005	0.09237	0.52837	-0.08697
0.54403	0.08819	0.55240	-0.08117
0.56801	0.08356	0.57663	-0.07442
0.59277	0.07872	0.60100	-0.06677
0.61747	0.07379	0.62649	-0.06112

0.64233	0.06889	0.65180	-0.05212
0.66718	0.06403	0.67710	-0.04792
0.69162	0.05917	0.70231	-0.04372
0.71606	0.05462	0.72752	-0.03558
0.73969	0.05017	0.75210	-0.02744
0.76314	0.04578	0.77668	-0.02466
0.78585	0.04138	0.80008	-0.02188
0.80756	0.03761	0.82348	-0.01559
0.82897	0.03384	0.84567	-0.00930
0.84854	0.03017	0.86677	-0.00859
0.86696	0.02650	0.88673	-0.00787
0.88537	0.02335	0.90545	-0.00370
0.90180	0.02020	0.92199	-0.00205
0.91763	0.01694	0.93852	-0.00075
0.93148	0.01368	0.95181	0.00017
0.94523	0.01101	0.96509	0.00054
0.95661	0.00834	0.97478	0.00112
0.96799	0.00600	0.98446	0.00065
0.97664	0.00418	0.99029	0.00029
0.98528	0.00245	0.99612	0.00024
0.99076	0.00145	0.99806	0.00013
0.99623	0.00054	1.00000	0.00000
0.99812	0.00026	-	-
1.00000	0.00000	-	-

B.2 NACA 0012 Airfoil

Figure B.2: NACA 0012 airfoil geometry.

Table B.2: Set of NACA 0012 airfoil coordinates provided by Harris (1981) with extra coordinates calculated in the present study using a piecewise cubic Hermite interpolating polynomial shaded in gray.

Upper Surface		Lower Surface	
x'/c	y'/c	x'/c	y'/c
0.00000	0.00000	0.00000	0.00000
0.00100	0.00757	0.00100	-0.00757
0.00200	0.00780	0.00200	-0.00780
0.00350	0.01008	0.00350	-0.01008
0.00500	0.01220	0.00500	-0.01220
0.00750	0.01400	0.00750	-0.01400
0.01000	0.01700	0.01000	-0.01700
0.01500	0.02050	0.01500	-0.02050
0.02000	0.02360	0.02000	-0.02360
0.02500	0.02500	0.02500	-0.02500

0.03000	0.02840	0.03000	-0.02840
0.03500	0.02930	0.03500	-0.03065
0.04000	0.03230	0.04000	-0.03230
0.04500	0.03320	0.04500	-0.03395
0.05000	0.03550	0.05000	-0.03550
0.05500	0.03640	0.05500	-0.03700
0.06000	0.03840	0.06000	-0.03840
0.06500	0.03930	0.06500	-0.04080
0.08000	0.04310	0.08000	-0.04310
0.08500	0.04400	0.08500	-0.04590
0.10000	0.04680	0.10000	-0.04680
0.10500	0.04770	0.10500	-0.04843
0.12000	0.04990	0.12000	-0.04990
0.13000	0.05120	0.13000	-0.05122
0.14000	0.05240	0.14000	-0.05240
0.15000	0.05340	0.15000	-0.05345
0.16000	0.05440	0.16000	-0.05440
0.17000	0.05530	0.17000	-0.05530
0.18000	0.05610	0.18000	-0.05610
0.19000	0.05670	0.19000	-0.05679
0.20000	0.05740	0.20000	-0.05740
0.21000	0.05800	0.21000	-0.05794
0.22000	0.05840	0.22000	-0.05840
0.23000	0.05870	0.23000	-0.05878
0.24000	0.05910	0.24000	-0.05910
0.25000	0.05950	0.25000	-0.05938
0.26000	0.05960	0.26000	-0.05960
0.27000	0.05970	0.27000	-0.05979
0.28000	0.05990	0.28000	-0.05990
0.29000	0.06000	0.29000	-0.06000
0.30000	0.06000	0.30000	-0.06000

0.31000	0.05997	0.31000	-0.05997
0.32000	0.05990	0.32000	-0.05990
0.33000	0.05982	0.33000	-0.05982
0.34000	0.05970	0.34000	-0.05970
0.35000	0.05953	0.35000	-0.05953
0.36000	0.05930	0.36000	-0.05930
0.37000	0.05902	0.37000	-0.05902
0.38000	0.05870	0.38000	-0.05870
0.39000	0.05836	0.39000	-0.05836
0.40000	0.05800	0.40000	-0.05800
0.41000	0.05761	0.41000	-0.05761
0.42000	0.05720	0.42000	-0.05720
0.43000	0.05676	0.43000	-0.05676
0.44000	0.05630	0.44000	-0.05630
0.45000	0.05581	0.45000	-0.05581
0.46000	0.05530	0.46000	-0.05530
0.47000	0.05477	0.47000	-0.05477
0.48000	0.05420	0.48000	-0.05420
0.49000	0.05356	0.49000	-0.05356
0.50000	0.05290	0.50000	-0.05290
0.51000	0.05226	0.51000	-0.05226
0.52000	0.05160	0.52000	-0.05160
0.53000	0.05091	0.53000	-0.05091
0.54000	0.05020	0.54000	-0.05020
0.55000	0.04951	0.55000	-0.04951
0.56000	0.04880	0.56000	-0.04880
0.57000	0.04809	0.57000	-0.04809
0.58000	0.04720	0.58000	-0.04720
0.59000	0.04640	0.59000	-0.04640
0.60000	0.04560	0.60000	-0.04560
0.61000	0.04481	0.61000	-0.04481

0.62000	0.04400	0.62000	-0.04400
0.63000	0.04380	0.63000	-0.04380
0.64000	0.04220	0.64000	-0.04220
0.65000	0.04100	0.65000	-0.04100
0.66000	0.04040	0.66000	-0.04040
0.67000	0.03920	0.67000	-0.03920
0.68000	0.03860	0.68000	-0.03860
0.69000	0.03740	0.69000	-0.03740
0.70000	0.03660	0.70000	-0.03660
0.71000	0.03540	0.71000	-0.03540
0.72000	0.03470	0.72000	-0.03470
0.73000	0.03390	0.73000	-0.03390
0.74000	0.03260	0.74000	-0.03260
0.75000	0.03140	0.75000	-0.03140
0.76000	0.03060	0.76000	-0.03060
0.77000	0.02940	0.77000	-0.02940
0.78000	0.02840	0.78000	-0.02840
0.79000	0.02720	0.79000	-0.02720
0.80000	0.02620	0.80000	-0.02620
0.81000	0.02510	0.81000	-0.02510
0.82000	0.02400	0.82000	-0.02400
0.83000	0.02280	0.83000	-0.02280
0.84000	0.02170	0.84000	-0.02170
0.85000	0.02050	0.85000	-0.02050
0.86000	0.01930	0.86000	-0.01930
0.87000	0.01810	0.87000	-0.01810
0.88000	0.01690	0.88000	-0.01690
0.89000	0.01570	0.89000	-0.01570
0.90000	0.01450	0.90000	-0.01450
0.91000	0.01330	0.91000	-0.01330
0.92000	0.01200	0.92000	-0.01200

0.93000	0.01073		
0.94000	0.00940	0.94000	-0.00940
0.95000	0.00807		
0.96000	0.00670	0.96000	-0.00670
0.97000	0.00537		
0.98000	0.00400	0.98000	-0.00400
0.99000	0.00267		
1.00000	0.00130	1.00000	-0.00130

References

- Abbott, I.H. & von Doenhoff, A.E. 1959, *Theory of Wing Sections: Including a Summary of Airfoil Data*, Dover Publications, New York, NY.
- Abu-Ghannam, B. & Shaw, R. 1980, "Natural transition of boundary layers - the effects of turbulence, pressure gradient and flow history", *Journal of Mechanical Engineering Science*, vol. 22, no. 5, pp. 213-228.
- Ainslie, J.F. 1987, "Calculating the flowfield in the wake of wind turbines", *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 27, no. 1-3, pp. 213-224.
- Anderson, J.D. 2011, *Fundamentals of aerodynamics*, 5th edn, McGraw-Hill, New York.
- Banks, W.H.H. & Gadd, G.E. 1963, "Delaying effects of rotation on laminar separation", *AIAA Journal*, vol. 1, no. 4, pp. 941-942.
- Berger, S.A. 1971, *Laminar wakes*, Elsevier, New York.
- Betz, A., 1926, *Wind-Energie und ihre ausnutzung durch Windmühlen*. Vandenhoeck, Göttingen.
- Bienen, T. & Von Karman, T. 1924, "The theory of air propellers (Zur Theorie der Luftschrauben)", *Zeitschrift des Vereines Deutscher Ingenieure*, vol. 68, no. 48, pp. 1237-1242.
- Blair, M.F. 1982, "Influence of free-stream turbulence on boundary layer transition in favourable pressure gradients", *Journal of Engineering for Power-Transactions of the ASME*, vol. 104, no. 4, pp. 743-750.
- Blasius, H. 1907, *Grenzschichten in Flüssigkeiten mit kleiner Reibung*, Druck von B. G. Teubner.
- Bothezat, G. 1919, *The general theory of blade screws*, NACA.
- Buhl, M.L. 2005, *A new empirical relationship between thrust coefficient and induction factor for the turbulent windmill state*, National Renewable Energy Laboratory, Golden, CO.
- Burton, T., Sharpe, D., Jenkins, N. & Bossanyi, E. 2001, *Wind energy handbook*, J. Wiley, Chichester, England.
- Carter, J.E. 1981, "Viscous-inviscid interaction analysis of transonic turbulent separated flow", *American Institute of Aeronautics and Astronautics, 14th Fluid and Plasma Dynamics Conference*, June 23-25, pp. 1.

- Catherall, D. & Mangler, K.W. 1966, "The integration of the two-dimensional laminar boundary-layer equations past the point of vanishing skin friction", *Journal of Fluid Mechanics*, vol. 26, pp. 163-182.
- Cebeci, T., Kafyeke, F., Laurendeau, E. & Shao, J.P. 2005, *Computational Fluid Dynamics for Engineers From Panel to Navier-Stokes Methods with Computer Programs*, Springer-Verlag GmbH, Berlin Heidelberg.
- Chamorro, L.P. & Porte-Agel, F. 2009, "A wind-tunnel investigation of wind-turbine wakes: boundary-layer turbulence effects", *Boundary-Layer Meteorology*, vol. 132, no. 1, pp. 129-149.
- Chandrasekhar, S. 1961, *Hydrodynamic and hydromagnetic stability*, Clarendon Press, Oxford.
- Chen, K.K. & Thyson, N.A. 1971, "Extension of Emmons' spot theory to flows on blunt bodies", *AIAA Journal*, vol. 9, pp. 821-825.
- Chong, T.P. & Zhong, S. 2005, "On the three-dimensional structure of turbulent spots", *Journal of Turbomachinery*, vol. 127, no. 3, pp. 545-551.
- Coles, D. 1964, "The turbulent boundary layer in a compressible fluid", *Physics of Fluids*, vol. 7, no. 9, pp. 1403-1424.
- Coles, D. 1956, "The law of the wake in the turbulent boundary layer", *Journal of Fluid Mechanics*, vol. 1, no. 02, pp. 191.
- Corrigan, J.J. & Schillings, J.J. 1994, "Empirical Model for Stall Delay due to Rotation", *Proceedings of the 1994 American Helicopter Society Aeromechanics Specialists Conference*, January 1994.
- Crespo, A., Manuel, F. & Hernandez, J. 1990, "Numerical modelling of wind turbine wakes", *European Community Wind Energy Conference*, , pp. 4-B4.
- Cummings, M.J. & Bragg, M.B. 1996, "Boundary-Layer Transition Due to Isolated Three-Dimensional Roughness on Airfoil Leading Edge (TN)", *AIAA Journal*, vol. 34, no. 9, pp. 1949-1952.
- Das, D.K. 1987, "Numerical study of turbulent separated flows", *Papers Presented at the 1987 ASME Applied Mechanics*, ASME, Bioengineering and Fluids Engineering Conference, pp. 85.
- Das, D.K. & White, F.M. 1986, "Integral skin friction prediction for turbulent separated flows", *Journal of Fluids Engineering*, vol. 108, no. 4, pp. 476-82.

- Datta, B.N. 2010, *Numerical linear algebra and applications*, 2nd edn, Society for Industrial and Applied Mathematics, Philadelphia.
- Davis, R.L., Carter, J.E. & Reshotko, E. 1987, "Analysis of transitional separation bubbles on infinite swept wings", *AIAA Journal*, vol. 25, no. 3, pp. 421-428.
- Dong, Y. & Cumpsty, N.A. 1990, "Compressor blade boundary layers. Part 2. Measurements with incident wakes", *Journal of Turbomachinery*, vol. 112, no. 2, pp. 231-240.
- Dossing, M. 2007, *Vortex lattice modelling of winglets on wind turbine blades*, Riso National Laboratory, Technical University of Denmark.
- D'Ovidio, A., Harkins, J.A. & Gostelow, J.P. 2001, *Turbulent spots in strong adverse pressure gradients - Part 2: Spot propagation and spreading rates*, ASME Paper no. 2001-GT-0406.
- Drela, M. & Giles, M.B. 1987, "ISES: a two-dimensional viscous aerodynamic design and analysis code", *AIAA 25th Aerospace Sciences Meeting*, pp. 1.
- Drzewiecki, S. 1920, *Théorie générale de l'hélice: hélices aériennes et hélices marines*, Gauthier-Villars et cie., Paris.
- Drzewiecki, S. 1892, *Méthode pour la détermination des éléments mécaniques des propulseurs hélicoidaux.*, Association Technique Maritime, Paris.
- Du, Z.H. & Selig, M.S. 1998, "A 3-D Stall Delay Model for HAWT Performance Prediction", *36th AIAA Aerospace Sciences Meeting and Exhibit and 1998 ASME Wind energy Symposium, Reno, NV, Jan. 12-15*, American Institute of Aeronautics and Astronautics, 1998.
- Dwyer, H.A. & McCroskey, W.J. 1971, "Crossflow and unsteady boundary- layer effects on rotating blades", *AIAA Journal*, vol. 9, no. 8, pp. 1498-1505.
- Emmons, H.W. 1951, "Laminar-turbulent transition in a boundary layer", *Journal of the Aeronautical Sciences*, vol. 18, no. 7, pp. 490-498.
- Erm, L.P., Smits, A.J. & Joubert, P.N. 1985, "Low Reynolds number turbulent boundary layers on a smooth flat surface in a zero pressure gradient", *Fifth Symposium on Turbulent Shear Flows.*, pp. 2. 13.
- Falkner, V.M. & Skan, S.W. 1931, "Solutions of the Boundary-layer equations in aerodynamics", *Philosophical Magazine*, vol. 12, pp. 865-896.
- Farin, G.E. 2002, *Curves and surfaces for CAGD : a practical guide*, 5th edn, Morgan Kaufmann, San Francisco, CA.

- Ferziger, J.H., Lyrio, A.A. & Bardina, J.G. 1982, "New skin friction and entrainment correlations for turbulent boundary layers", *Journal of Fluids Engineering, Transactions of the ASME*, vol. 104, no. 4, pp. 537-40.
- Fletcher, T.M., Brown, R.E., Kim, R.E. & Kwon, O.J. 2009, "Predicting wind turbine blade loads using vorticity transport and RANS methodologies", *European Wind Energy Conference 2009*, March 2009.
- Frankl, F.I. 1934, , Transactions of the Central Aero-Hydrodynamics Institute, Moscow.
- Fraser, C.J., Higazy, M.G. & Milne, J.S. 1994, "End-stage boundary layer transition models for engineering calculations", *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, vol. 208, no. 1, pp. 47-58.
- Gaster, M. 1969, *The structure and behaviour of laminar separation bubbles*, H.M.S.O., London.
- George, W.K. 1995, "Some new ideas for similarity of turbulent shear flows" in *Proc. ICHMT Symposium on Turbulence, Heat and Mass Transfer, Lisbon, Portugal (1994)*, eds. K. Hanjalic & J. Pereira, Begell House, .
- George, W.K. 1989, "The self-preservation of turbulent flows and its relation to initial conditions and coherent structures" in *Advances in turbulence*, eds. W.K. George & R. Arndt, Hemisphere, New York, NY, pp. 39-73.
- George, W.K. & Castillo, L. 1997, "Zero-pressure-gradient turbulent boundary layer", *Applied Mechanics Reviews*, vol. 50, no. 12, pp. 689-729.
- Gilmer, B.R. & Bristow, D.R. 1982, "Analysis of stalled airfoils by simultaneous perturbations to viscous and inviscid equations", *AIAA Journal*, vol. 20, no. 9, pp. 1160-6.
- Gilmer, B.R., Jasper, D.W. & Bristow, D.R. 1984, "Analysis of stalled multi-element airfoils", *AIAA 2nd Applied Aerodynamics Conference.*, AIAA, pp. AIAA, New York, NY, USA.
- Glauert, H. 1983, *The elements of aerofoil and airscrew theory*, Cambridge University Press, Cambridge [Cambridgeshire]; New York.
- Glauert, H. 1935, "Airplane Propellers" in *Aerodynamic Theory*, ed. W.F. Durand, Julius Springer, Berlin, pp. 169-360.
- Glauert, H. 1926, *The analysis of experimental results in the windmill brake and vortex ring states of an airscrew*, Aeronautical Research Council, London.

- Glauert, H. 1922, *An Aerodynamic Theory of the Airscrew*, Aeronautical Research Council, London.
- Goldstein, S. 1948, "On laminar boundary-layer flow near a position of separation", *Quarterly Journal of Mechanics and Applied Mathematics*, vol. 1, pp. 43-69.
- Goldstein, S. 1929, "Vortex theory of screw propellers", *Proceedings of the Royal Society of London*, vol. 123, pp. 440-465.
- Görtler, H. 1942, "Berechnung von Aufgaben der freien Turbulenz auf Grund eines neuen Näherungsansatzes .", *Zeitschrift für Angewandte Mathematik und Mechanik*, vol. 22, no. 5, pp. 244-254.
- Gostelow, J.P. & Blunden, A.R. 1989, "Investigations of boundary layer transition in an adverse pressure gradient", *Journal of Turbomachinery*, vol. 111, no. 4, pp. 366-375.
- Gostelow, J.P., Blunden, A.R. & Walker, G.J. 1994, "Effects of free-stream turbulence and adverse pressure gradients on boundary layer transition", *Journal of Turbomachinery*, vol. 116, no. 3, pp. 392-404.
- Gostelow, J.P., Melwani, N. & Walker, G.J. 1996, "Effects of streamwise pressure gradient on turbulent spot development", *Journal of Turbomachinery*, vol. 118, pp. 737-743.
- Gostelow, J.P. & Thomas, R.L. 2005, "Response of a laminar separation bubble to an impinging wake", *Journal of Turbomachinery*, vol. 127, no. 1, pp. 35-42.
- Gowda, B.H.L. & Narayana, P.A.A. 1980, "An experimental investigation of separating flow on a convex surface", *Applied Scientific Research*, vol. 36, no. 4, pp. 271-88.
- Hall, A.A. & Hislop, G.S. 1938, *Experiments on the transition of the laminar boundary layer on a flat plate*, Aeronautical Research Council, London.
- Hand, M.M., Simms, D.A., Fingersh, L.J., Jager, D.W., Cotrell, J.R. & Schreck, S.J. 2001, *Unsteady Aerodynamics Experiment Phase VI: Wind Tunnel Test Configurations and Available Data Campaigns*, National Renewable Energy Laboratory, Golden, CO.
- Harris, C.D. 1981, *Two-dimensional aerodynamic characteristics of the NACA 0012 airfoil in the Langley 8-foot transonic pressure tunnel*, NASA Langley Research Center, Hampton, VA.
- Hartnett, J.P. & Eckert, E.R.G. 1957, "Mass-transfer cooling in laminar boundary layer with constant fluid properties", *American Society of Mechanical Engineers - Transactions*, vol. 79, no. 2, pp. 247-254.

- Hartree, D.R. 1937, "On an equation occurring in Falkner and Skan's approximate treatment of the equations of the boundary layer", *Mathematical Proceedings of the Cambridge Philosophical Society*, vol. 33, no. 2.
- Hatman, A. & Wang, T. 1999, "A prediction model for separated-flow transition", *Journal of Turbomachinery*, vol. 121, no. 3, pp. 594-602.
- Hau, E. 2006, *Wind turbines fundamentals, technologies, application, economics*, 2nd [English] edn, Springer, Berlin ; New York.
- Hayes, W.D. & Probstein, R.F. 1959, *Hypersonic flow theory*, 1st edn, Academic Press, New York.
- Head, M.R. 1958, *Entrainment in the Turbulent Boundary Layer*, Aeronautical Research Council, London.
- Hess, J. 1990, "Panel Methods In Computational Fluid Dynamics", *Annual Review of Fluid Mechanics*, vol. 22, no. 1, pp. 255-274.
- Hess, J.L. & Smith, A.M.O. 1967, "Calculation of potential flow about arbitrary bodies", *Progress in Aeronautical Sciences*, vol. 8, pp. viii + 431-138.
- Hess, J.L. & Smith, A.M.O. 1964, "Calculation of nonlifting potential flow about arbitrary three-dimensional bodies", *Journal of Ship Research*, vol. 8, no. 2, pp. 22-44.
- Himmelskamp, H. 1950, *Profiluntersuchungen an einem Umlaufenden Propeller (Profile Investigations on a Rotating Propeller)*, Max-Planck Institut für Stromungsforschung, Göttingen, Germany.
- Himmelskamp, H. 1947, *Profile investigations on a rotating airscrew*, MAP Völkenrode Reports and Translation No. 832.
- Högstrom, U., Asimakopoulos, D.N., Kambezidis, H., Helmis, C.G. & Smedman, A. 1988, "Field study of the wake behind a 2 MW wind turbine", *Atmospheric Environment - Part A General Topics*, vol. 22, no. 4, pp. 803-820.
- Hojstrup, J. 1999, "Spectral coherence in wind turbine wakes", *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 80, no. 1-2, pp. 137-146.
- Horton, H.P. 1969, *A semi-empirical theory for the growth and bursting of laminar separation bubbles*, Aeronautical Research Council, London.
- Hourmouziadis, J. 1989, "Aerodynamic design of low pressure turbines", *Blading Design for Axial Turbomachines - AGARD Lecture Series*, AGARD, pp. 8.1.

- Howarth, L. 1938, "Solution of the laminar boundary layer equations", *Proceedings of the Royal Society of London. Ser. A*, vol. 164A, pp. 547-579.
- Huyer, S.A., Simms, D.A. & Robinson, M.C. 1996, "Unsteady aerodynamics associated with a horizontal-axis wind turbine", *AIAA Journal*, vol. 34, no. 7, pp. 1410-19.
- Iglisch, R. 1944, "Exakte Berechnung der laminaren Grenzschicht an der längsangeströmten ebenen Platte mit homogener Absaugung", *Schriften der Deutschen Akademie der Luftfahrtforschung. Ser. B*, , pp. 1-51.
- Johnson, M.W. & Fashifar, A. 1994, "Statistical properties of turbulent bursts in transitional boundary layers", *International Journal of Heat and Fluid Flow*, vol. 15, no. 4, pp. 283-90.
- Johnson, M.W. & Ercan, A.H. 1997, "Predicting bypass transition: A physical model versus empirical correlations", *Proceedings of the 1997 International Gas Turbine & Aeroengine Congress & Exposition, June 2, 1997 - June 5, ASME, 1997*, pp. ASME.
- Katz, J. & Plotkin, A. 1991, *Low-speed aerodynamics: from wing theory to panel methods*, McGraw-Hill, New York.
- Kim, J., Simon, T.W. & Russ, S.G. 1992, "Free-stream turbulence and concave curvature effects on heated, transitional boundary layers", *Journal of Heat Transfer*, vol. 114, no. 2, pp. 338-347.
- Kline, S.J., Cantwell, B.J. & Lilley, G.M. 1981, "The 1980-81 AFOSR-HTTM Stanford Conference on Complex Turbulent Flows: Comparison of Computation and Experiment", Mechanical Engineering Dept., Stanford University, September 3-6, 1980, September 14-18, 1981.
- Kuethe, A.M. & Chow, C. 1998, *Foundations of aerodynamics : bases of aerodynamic design*, 5th edn, J. Wiley, New York.
- Lagerstrom, P.A. 1975, "Solutions of the Navier-Stokes equation at large Reynolds number", *SIAM Journal on Applied Mathematics*, vol. 28, no. 1, pp. 202-14.
- Lagrée, P.Y. 2009, *Notes on triple deck*, Paris, France.
- Lanchester, F.W. 1915, "A contribution to the theory of propulsion and the screw propeller", *Journal of the American Society for Naval Engineers*, vol. 27, no. 2, pp. 509-510.
- LeBalleur, J.C. 1978, "Couplage visqueux-non visqueux: méthode numérique et applications aux écoulements bidimensionnels transsoniques et supersoniques.", *La Recherche Aérospatiale*, vol. 183, pp. 65-76.

- Leibenson, L.S. 1935, "Energy form of integral condition in theory of boundary layer", *Doklady Akademiyi Nauk U.S.S.R.*, vol. 2, no. 1, pp. 22-24.
- Leishman, J.G. 2002, "Challenges in modeling the unsteady aerodynamics of wind turbines", *ASME 2002 Wind Energy Symposium, WIND2002*, Web Portal ASME (American Society of Mechanical Engineers), January 14-17, 2002, pp. 141.
- Leishman, J.G. 2006, *Principles of helicopter aerodynamics*, Cambridge University Press, Cambridge.
- Leonard, A. 1980, "Vortex methods for flow simulation", *Journal of Computational Physics*, vol. 37, no. 3, pp. 289-335.
- Liepmann, H.W. 1943, *Investigations on laminar boundary-layer stability and transition on curved boundaries*, National Advisory Committee for Aeronautics, Washington, D.C.
- Lighthill, M.J. 2000, "Upstream influence in boundary layers 45 years ago", *Philosophical Transactions of the Royal Society London, Ser. A*, vol. 358, no. 1777, pp. 3047-61.
- Lighthill, M.J. 1953, "On boundary layers and upstream influence. I. A comparison between subsonic and supersonic flows", *Proceedings of the Royal Society of London. Ser. A*, vol. 217, pp. 344-357.
- Lindenburg, C. 2004, "Modelling of rotational augmentation based on engineering considerations and measurements", *Proceedings of the 2004 European Wind Energy Conference & Exhibition*, 22nd-25th November 2004.
- Lissaman, P.B.S. 1979, "Energy effectiveness of arbitrary arrays of wind turbines", *Journal of Energy*, vol. 3, no. 6, pp. 323-8.
- Lock, C.N.H., Bateman, H. & Townend, H. 1925, *An extension of the vortex theory of airscrews with applications to airscrews of small pitch and including experimental results*, Aeronautical Research Council, London.
- Lock, R.C. 1951, "The velocity distribution in the laminar boundary layer between parallel streams", *The Quarterly Journal of Mechanics and Applied Mathematics*, vol. 4, no. 1, pp. 42-63.
- Ludwig, H. & Tillmann, W. 1950, *Investigations of the wall-shearing stress in turbulent boundary layers (Untersuchungen über die Wandschubspannung in turbulenten Reibungsschichten)*, National Advisory Committee for Aeronautics, Washington, D.C.

- Mack, L.M. 1984, *Boundary-layer linear stability theory*, Ft. Belvoir Defense Technical Information Center, Ft. Belvoir, VA.
- Madsen, H.A. & Christensen, H.F. 1990, "On the relative importance of rotational, unsteady and three-dimensional effects on the HAWT rotor aerodynamics", *Wind Engineering*, vol. 14, no. 6, pp. 405-15.
- Maeda, T., Kamada, Y., Murata, J., Yonekura, S., Ito, T., Okawa, A. & Kogaki, T. 2011, "Wind tunnel study on wind and turbulence intensity profiles in wind turbine wake", *Journal of Thermal Science*, vol. 20, no. 2, pp. 127-32.
- Manwell, J.F., McGowan, J.G. & Rogers, A.L. 2009, *Wind energy explained : theory, design and application*, 2nd edn, Wiley, Chichester, U.K.
- Martin, B.W., Brown, A. & Garrett, S.E. 1978, "Heat transfer to a PVD rotor blade at high subsonic passage throat Mach numbers", *Heat and Fluid Flow*, vol. 8, no. 1, pp. 33-43.
- Mayle, R.E. 1991, "Role of laminar-turbulent transition in gas turbine engines", *Journal of Turbomachinery*, vol. 113, pp. 509-537.
- McCormick, B.W. 1967, *Aerodynamics of V/STOL flight*, Academic Press, New York, NY.
- Messiter, A.F. 1970, "Boundary-layer flow near the trailing edge of a flat plate", *SIAM Journal on Applied Mathematics*, vol. 18, no. 1, pp. 241-57.
- Milborrow, D.J. 1985, "Changes in aerofoil characteristics due to radial flow on rotating blades", *Proceedings of the 7-th BWEA conference*, ed. A. Garrad.
- Morkovin, M.V. 1969, "On the many faces of transition" in *Viscous drag reduction*, ed. C.S. Wells, Platinum Press, New York, NY, pp. 1-31.
- Moses, H.L., Jones III, R.R., O'Brien Jr., W.F. & Peterson, R.S. 1978, "Simultaneous solution of the boundary layer and freestream with separated flow", *AIAA Journal*, vol. 16, no. 1, pp. 61-66.
- Murlis, J., Tsai, H.M. & Bradshaw, P. 1982, "The structure of turbulent boundary layers at low Reynolds numbers", *Journal of Fluid Mechanics*, vol. 122, pp. 13-56.
- Narasimha, R. 1957, "On the distribution of intermittency in the transition region of a boundary layer", *Journal of Aeronautical Sciences*, vol. 24, pp. 711-712.
- Narramore, J.C. & Vermeland, R. 1992, "Navier-Stokes calculations of inboard stall delay due to rotation", *Journal of Aircraft*, vol. 29, no. 1, pp. 73-78.

- Neiland, V.Y. 1969, "Theory of the separation of a laminar boundary layer in a supersonic stream", *Izvestiya Akademii Nauk SSSR, Mekhanika Zhidkosti i Gaza*, , no. 4, pp. 53-7.
- Pai Shih-I 1954, *Fluid dynamics of jets*, Van Nostrand, New York, NY.
- Pinson, M.W. & Wang, T. 1997, "Effects of leading-edge roughness on fluid flow and heat transfer in the transitional boundary layer over a flat plate", *International Journal of Heat and Mass Transfer*, vol. 40, no. 12, pp. 2813-2823.
- Pope, S.B. 2000, *Turbulent flows*, Cambridge University Press, Cambridge, NY.
- Powell, M.J.D. 1968, *A FORTRAN subroutine for solving systems of non-linear algebraic equations*, United Kingdom Atomic Energy Authority, Research Group, A.E.R.E., Harwell, Berkshire, UK.
- Prandtl, L. 1905, "Über Flüssigkeitsbewegung mit kleiner Reibung.", *Verhandlungen des III. internationalen Mathematischen Kongresses*, Teubner, 1904, pp. 484.
- Prandtl, L. & Betz, A. 1919, *Schraubenpropeller mit geringstem Energieverlust*, Göttinger Nachrichten, Göttingen.
- Prandtl, L. 1921, "Bemerkungen über die Entstehung der Turbulenz", *Zeitschrift für Angewandte Mathematik und Mechanik*, vol. 1, no. 6, pp. 431-436.
- Prandtl, L. 1918, *Tragflügeltheorie.*, Göttingen.
- Press, W.H., Teukolsky, S.A., Vetterling, W.T. & Flannery, B.P. 1988, *Numerical recipes in C: the art of scientific computing*, Cambridge University Press, Cambridge, NY.
- Purtell, L.P., Klebanoff, P.S. & Buckley, F.T. 1981, "Turbulent boundary layer at low Reynolds number", *Physics of Fluids*, vol. 24, no. 5, pp. 802-11.
- Ramsay, R.R., Hoffmann, M.J. & Gregorek, G.M. 1995, *Effects of Grit Roughness and Pitch Oscillations on the S809 Airfoil*, National Renewable Energy Laboratory, Golden, CO.
- Reissner, H. 1940, *Generalized Vortex theory of screw propeller and its application*, National Advisory Committee for Aeronautics (United States Advisory Committee for Aeronautics), Washington, DC, United States.
- Reissner, H. 1937, "Vortex theory of screw propeller", *Journal of the Aeronautical Sciences*, vol. 5, no. 1, pp. 1-7.
- Reissner, H. 1910, *Z. Flugtech. Motorluftschiffahrt*, vol. 1, pp. 257-309.

- Roberts, S.K. & Yaras, M.I. 2005a, "Boundary-layer transition affected by surface roughness and free-stream turbulence", *Journal of Fluids Engineering, Transactions of the ASME*, vol. 127, no. 3, pp. 449-457.
- Roberts, S.K. & Yaras, M.I. 2005b, "Modeling transition in separated and attached boundary layers", *Journal of Turbomachinery*, vol. 127, no. 2, pp. 402-11.
- Roberts, S.K. & Yaras, M.I. 2003, "Measurements and prediction of free-stream turbulence and pressure-gradient effects on attached-flow boundary-layer transition", *2003 ASME Turbo Expo, June 16, 2003 - June 19, American Society of Mechanical Engineers*, 2003, pp. 729.
- Roberts, S.K. 2005, *Boundary layer transition in attached and separated flows at low Reynolds numbers*, Ph.D. thesis, Carleton University (Canada).
- Roberts, W.B. 1980, "Calculation of laminar separation bubbles and their effect on airfoil performances", *AIAA Journal*, vol. 18, no. 1, pp. 26-31.
- Roberts, W.B. 1975, "The effect of Reynolds number and laminar separation on axial cascade performance", *Journal of Engineering for Power, Transactions ASME*, vol. 97 Ser A, no. 2, pp. 261-274.
- Robinson, M.C., Galbraith, R.A.M., Shipley, D. & Miller, M. 1995, "Unsteady aerodynamics of wind turbines", *14th ASME Wind ENergy Symposium and the 33rd Aerospace Sciences Meeting, Reno, NV, AIAA*, Janary 9-12, 1995.
- Robinson, M.C., Simms, D.A., Hand, M.M. & Schreck, S.J. 1999, "Horizontal Axis Wind Turbine Aerodynamics: Three Dimensional, Unsteady and Separated Flow Influences", *3rd ASME/JSME Joint Fluids Engineering Conference, San Francisco, CA, July 18-23*.
- Rothmayer, A.P. & Barnett, M. 1991, "Constructing physics-based numerical schemes for viscous-inviscid interactions", *Computers and Fluids*, vol. 20, no. 3, pp. 201-211.
- Sarpkaya, T. 1989, "Computational methods with vortices - the 1988 Freeman Scholar Lecture", *Journal of Fluids Engineering, Transactions of the ASME*, vol. 111, no. 1, pp. 5-52.
- Savill, A.M. 1993a, "Evaluating turbulence model predictions of transition", *Applied Scientific Research (The Hague)*, vol. 51, no. 1-2, pp. 555-562.
- Savill, A.M. 1993b, "Some recent progress in the turbulence modelling of by-pass transition", *Proceedings of the International Conference on Near- Wall Turbulent Flows*, Publ by Elsevier Science Publishers B.V, Mar 15 - 17 1993, pp. 829.

- Schlichting, H. 1933, "Laminare Strahlausbreitung", *Zeitschrift für Angewandte Mathematik und Mechanik*, vol. 13, no. 4, pp. 260-263.
- Schlichting, H. & Bussmann, K. 1943, "Exakte Lösungen für die laminare Grenzschicht mit Absaugung und Ausblasen", *Schriften der Deutschen Akademie der Luftfahrtforschung. Ser. B*, vol. 7, no. 2.
- Schlichting, H. & Gersten, K. 2000, *Boundary-layer theory (Grenzschicht-theorie)*, 8th edn, Springer, Berlin; New York.
- Schmid, P.J. & Henningson, D.S. 2001, *Stability and transition in shear flows*, Springer, New York.
- Schobeiri, M.T. & Radke, R.E. 1994, "Effects of periodic unsteady wake flow and pressure gradient on boundary layer transition along the concave surface of a curved plate", *Proceedings of the International Gas Turbine and Aeroengine Congress and Exposition, June 13, 1994 - June 16*, Publ by ASME, 1994, pp. 1.
- Schreck, S.J., Robinson, M.C., Hand, M.M. & Simms, D.A. 2000, "HAWT Dynamic Stall Response Asymmetries Under Yawed Flow Conditions", *19th ASME Wind Energy Symposium and the 38th AIAA Aerospace Sciences Meeting, Reno, NV*, AIAA, Jan 10-13.
- Schreck, S.J., Robinson, M.C., Hand, M.M. & Simms, D.A. 2001, "Blade dynamic stall vortex kinematics for a horizontal axis wind turbine in yawed conditions", *Journal of Solar Energy Engineering*, vol. 123, no. 4, pp. 272-81.
- Schubauer, G.B. & Klebanoff, P.S. 1955, *Contributions on mechanics of boundary-layer transition*, National Advisory Committee for Aeronautics, Washington, DC, United States.
- Schubauer, G.R. & Skramstad, H.K. 1948, *Laminar-boundary-layer oscillations and transition on flat plate*, National Advisory Committee for Aeronautics, Washington, DC.
- Schultz, M.P. & Volino, R.J. 2003, "Effects of concave curvature on boundary layer transition under high freestream turbulence conditions", *Journal of Fluids Engineering, Transactions of the ASME*, vol. 125, no. 1, pp. 18-27.
- Sezer-Uzol, N. & Uzol, O. 2009, "Effect of steady and transient wind shear on the wake structure and performance of a horizontal axis wind turbine rotor", *47th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, January 5, 2009 - January 8*, American Institute of Aeronautics and Astronautics, 2009.

- Sharma, O.P., Wells, R.A., Schlinker, R.H. & Bailey, D.A. 1982, "Boundary layer development on turbine airfoil suction surfaces", *Journal of Engineering for Power-Transactions of the ASME*, vol. 104, no. 3, pp. 698-706.
- Simms, D.A., Schreck, S.J., Hand, M.M. & Fingersh, L.J. 2001, *NREL unsteady aerodynamics experiment in the NASA-Ames wind tunnel: a comparison of predictions to measurements*, National Renewable Energy Laboratory, Golden, CO.
- Singh, N.K. & Sarkar, S. 2011, "DNS of a laminar separation bubble", *World Academy of Science, Engineering and Technology*, vol. 81, pp. 439-443.
- Snel, H. 1991, "Scaling Laws for the Boundary Layer Flow on Rotating Wind Turbine Blades", *4th IEA Symposium on Aerodynamics for Wind Turbines*, Rome, Nov. 20-21.
- Snel, H., Houwink, R. & Bosschers, J. 1994, *Sectional prediction of lift coefficients on rotating wind turbine blades in stall*, Netherlands Energy Research Foundation, Petten, Netherlands.
- Solomon, W.J., Walker, G.J. & Gostelow, J.P. 1995, "Transition length prediction for flows with rapidly changing pressure gradients", *Proceedings of the International Gas Turbine and Aeroengine Congress and Exposition, June 5, 1995 - June 8, ASME*, 1995, pp. 10.
- Somers, D.M. 1997, *Design and experimental results for the S809 airfoil*, National Renewable Energy Laboratory, Golden, CO.
- Sorensen, N.N., Michelsen, J.A. & Schreck, S.J. 2002, "Navier-Stokes Predictions of the NREL Phase VI Rotor in the NASA Ames 80-By-120 Wind Tunnel", *ASME Conference Proceedings*, vol. 2002, no. 7476X, pp. 94-105.
- Spalart, P.R. 1988, "Direct simulation of a turbulent boundary layer up to $R=1410$ ", *Journal of Fluid Mechanics*, vol. 187, pp. 61-98.
- Sreenivasan, K.R. & Narasimha, R. 1982, "Equilibrium parameters for two-dimensional turbulent wakes", *Transactions of the ASME. Journal of Fluids Engineering*, vol. 104, no. 2, pp. 167-70.
- Stewartson, K. 1968, "On the Flow Near the Trailing Edge of a Flat Plate", *Proceedings of the Royal Society of London. Ser. A*, vol. 306, no. 1486, pp. 275-290.
- Stieger, R.D. & Hodson, H.P. 2004, "The transition mechanism of highly loaded low-pressure turbine blades", *Journal of Turbomachinery*, vol. 126, no. 4, pp. 536-543.
- Tangler, J.L. & Selig, M.S. 1997, *An evaluation of an empirical model for stall delay due to rotation for HAWTs*, National Renewable Energy Laboratory, Golden, CO.

- Theodorsen, T. 1948, *Theory of propellers*, McGraw-Hill, New York, NY.
- Thwaites, B. 1949, "Approximate calculation of laminar boundary layer", *Aeronautical Quarterly*, vol. 1, pp. 245-280.
- Tokaty, G.A. 1971, *A history and philosophy of fluid mechanics*, Foulis, Henley-on-Thames.
- Tollmien, W. 1931, "Grenzschichttheorie" in *Handbuch der Experimentalphysik* Akad. Verlagsanstalt, Leipzig, pp. 241-287.
- Tollmien, W. 1929, "Über die Entstehung der Turbulenz", *Nachrichten von der Gesellschaft der Wissenschaften zu Göttingen*, vol. 11, pp. 407-409.
- Umur, H. & Ozalp, A.A. 2006, "Fluid flow and heat transfer in transitional boundary layers: effects of surface curvature and free stream velocity", *Heat and Mass Transfer*, vol. 43, no. 1, pp. 7-15.
- Veldman, A.E.P. 2009, "A simple interaction law for viscous-inviscid interaction", *Journal of Engineering Mathematics*, vol. 65, no. 4, pp. 367-383.
- Veldman, A.E.P. 2005, "Quasi-simultaneous viscous-inviscid interaction for transonic airfoil flow", *4th AIAA Theoretical Fluid Mechanics Meeting*, pp. 1.
- Veldman, A.E.P. 2001, "Matched asymptotic expansions and the numerical treatment of viscous-inviscid interaction", *Journal of Engineering Mathematics*, vol. 39, no. 1-4, pp. 189-206.
- Veldman, A.E.P. 1981, "New, quasi-simultaneous method to calculate interacting boundary layers", *AIAA Journal*, vol. 19, no. 1, pp. 79-91.
- Veldman, A.E.P. 1979, *A numerical method for the calculation of laminar, incompressible boundary layers with strong viscous-inviscid interaction*, Nationaal Lucht -en Ruimtevaartlaboratorium, Verslagen en Verhandelingen, Amsterdam, Netherlands.
- Veldman, A.E.P. & Coenen, E.G.M. 2005, *Interaction Laws in Viscous-Inviscid Coupling*, University of Groningen, Research Institute for Mathematics and Computing Science (IWI).
- Veldman, A.E.P., Lindhout, J.P.F., de Boer, E. & Somers, M.A.M. 1990, "Vistras: a simulation method for strongly-interacting viscous transonic flow" in *Numerical and physical aspects of aerodynamic flow*, ed. T. Cebeci, 4th edn, Springer, Berlin, pp. 37-51.

- Vermeer, L.J., Srensen, J.N. & Crespo, A. 2003, "Wind turbine wake aerodynamics", *Progress in Aerospace Sciences*, vol. 39, no. 6-7, pp. 467-510.
- Viterna, L.A. & Corrigan, R.D. 1982, *Fixed pitch rotor performance of large horizontal axis wind turbines*, NASA Lewis Research Center, Cleveland, OH.
- Volino, R.J. & Hultgren, L.S. 2001, "Measurements in separated and transitional boundary layers under low-pressure turbine airfoil conditions", *Journal of Turbomachinery*, vol. 123, no. 2, pp. 189-197.
- Volino, R.J. 2002, "Separated flow transition under simulated low-pressure turbine airfoil conditions - Part 1: Mean flow and turbulence statistics", *Journal of Turbomachinery*, vol. 124, no. 4, pp. 645-655.
- von Karman, T. 1921, "Über laminare und turbulente Reibung (Laminar and turbulent friction)", *Zeitschrift für Angewandte Mathematik und Mechanik*, vol. 1, no. 4, pp. 233-252.
- Walker, J.F. & Jenkins, N. 1997, *Wind energy technology*, John Wiley, New York.
- White, F.M. 2011, *Fluid mechanics*, 7th edn, McGraw Hill, New York, N.Y.
- White, F.M. 2006, *Viscous fluid flow*, 3rd edn, McGraw-Hill Higher Education, New York, NY.
- Wilson, R.E., Lissaman, P.B.S. & Walker, S.N. 1976, *Aerodynamic performance of wind turbines*, Oregon State University, Corvallis, OR.
- Wynanski, I., Champagne, F. & Marasli, B. 1986, "On the large-scale structures in two-dimensional, small-deficit, turbulent wakes", *Journal of Fluid Mechanics*, vol. 168, pp. 31-71.
- Yaras, M.I. 2002, "Measurements of the effects of freestream turbulence on separation-bubble transition", *ASME TURBO EXPO 2002: Heat Transfer, Manufacturing Materials and Metallurgy, June 3, 2002 - June 6*, American Society of Mechanical Engineers, 2002, pp. 647.
- Young, W.H., Jr. & Williams, J.C. 1972, "Boundary-layer separation on rotating blades in forward flight", *AIAA Journal*, vol. 10, no. 12, pp. 1613-19.
- Zhong, S., Chong, T.P. & Hodson, H.P. 2002, "On the spreading angle of turbulent spots in non-isothermal boundary layers with favourable pressure gradients", *ASME TURBO EXPO 2002: Heat Transfer, Manufacturing Materials and Metallurgy, June 3, 2002 - June 6*, American Society of Mechanical Engineers, 2002, pp. 571.