Improvement of bem analysis to incorporate stall delay effect and the study of atmospheric boundary layer effect on the wake characteristics of NREL phase VI turbine

Ijaz Fazil Syed Ahmed Kabir

2018

Ijaz Fazil Syed Ahmed Kabir. (2018). Improvement of bem analysis to incorporate stall delay effect and the study of atmospheric boundary layer effect on the wake characteristics of NREL phase VI turbine. Doctoral thesis, Nanyang Technological University, Singapore.

<http://hdl.handle.net/10356/75824>

<https://doi.org/10.32657/10356/75824>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

IMPROVEMENT OF BEM ANALYSIS TO INCORPORATE STALL DELAY EFFECT AND
THE STUDY OF ATMOSPHERIC BOUNDARY LAYER EFFECT ON THE WAKE

IJAZ FAZIL SYED AHMED KABIR 2018

**IMPROVEMENT OF BEM ANALYSIS TO INCORPORATE
STALL DELAY EFFECT AND THE STUDY OF
ATMOSPHERIC BOUNDARY LAYER EFFECT ON THE
WAKE CHARACTERISTICS OF NREL PHASE VI TURBINE**

IJAZ FAZIL SYED AHMED KABIR

SCHOOL OF MECHANICAL AND AEROSPACE ENGINEERING

2018

**IMPROVEMENT OF BEM ANALYSIS TO INCORPORATE
STALL DELAY EFFECT AND THE STUDY OF
ATMOSPHERIC BOUNDARY LAYER EFFECT ON THE
WAKE CHARACTERISTICS OF NREL PHASE VI TURBINE**

IJAZ FAZIL SYED AHMED KABIR

School of Mechanical and Aerospace Engineering

A thesis submitted to Nanyang Technological University in partial fulfilment of the
requirement for the degree of Doctor of Philosophy

2018

ACKNOWLEDGEMENTS

First, I show my heartiest gratitude to the most merciful ALLAH who has given me the strength and ability to write this thesis; without his help, it would have been impossible to end my project and write this doctoral thesis.

Then, I would like to express my deepest sense of gratitude and indebtedness to my supervisor, **Associate Professor Eddie Y.K. Ng**, for giving me the opportunity to work on this project under his guidance. In particular, the suggestions and recommendations of my supervisor during the course of this research work and his support during my personnel difficult situations have been invaluable. I would like to thank Nanyang Technological University for the financial support throughout my candidature.

I am also indebted to Dr. Teymour Javaherchi (Post-Doc, University of Washington), Dr. Mohammad Moshfeghi (Post-Doc, Sogang University), Dr. Ahmed Al Makky (Post-Doc, University of the West of Scotland), Mr. Ferozkhan Safiyullah (Research student, Universiti Teknologi Petronas) for providing me opportunities to discuss the technical issues and clarify my doubts.

I wish to extend my appreciation to all the technical staffs of CAE Lab 1 for their warm relationship and kind cooperation.

I acknowledge my thanks to all the members of Prof. Ng's research team: Mr. Koh Wei Xiang Martin, Mr. Chew Kok Hon, Mr. Koh Jian Hao and Mr. AbdulAziz Singapore Wala.

I'm also indebted to my friends in NTU particularly Mr. Roomi Mustafa, Mr. Mustafa Khalid, Mr. Mohd Tariq, Mr. Sivamoorthy Kanagalingam and Mr. Mohankumar for their support.

Last but not least, I record my heartfelt thanks to my respected parents, my beloved wife, my blessed son and all other family members for their understanding, kindness and encouragement during my course of study.

TABLE OF CONTENTS

ACKNOWLEDGEMENTS.....	i
TABLE OF CONTENTS	iii
LIST OF FIGURES.....	vii
LIST OF TABLES.....	xiii
ACRONYMS.....	xiv
NOMENCLATURE.....	xvi
SUMMARY	xxiv
CHAPTER 1: INTRODUCTION.....	1
1.1. Energy need in the world.....	1
1.2. Energy resources in the world	1
1.2.1. Conventional energy.....	1
1.2.2. Renewable energy	2
1.3. Historical development of wind turbines	2
1.4. Recent development of wind turbines	3
1.5. Aerofoil used in wind turbine blades.....	4
1.5.1. Preamble.....	4
1.5.2. Aerodynamics of Aerofoil	4
1.6. Aim of this study	6
1.7. Original contributions of the present study	7
1.8. Limitations of the present study.....	7
1.9. Publications based on current work	7
1.9.1. Journals.....	7
1.10. Outline of this thesis report.....	8
CHAPTER 2: AERODYNAMICS OF WIND TURBINES.....	9
2.1. Introduction	9
2.2. Actuator Disk Model	9
2.3. Momentum Theory.....	13
2.4. Blade Element Theory.....	16
2.5. Blade Element Momentum (BEM) Theory.....	18
2.6. Correction Factors for BEM Theory	19
2.6.1. Prandtl Tip Loss Correction.....	19
2.6.2. Glauert Correction	20
2.6.3. Hub Loss Model	21

2.6.4. Buhl Model.....	21
2.7. BEM algorithm	22
2.8. Unsteady Aerodynamics and 3D Aspects	23
2.9. Stall delay or Rotational Augmentation.....	26
2.9.1. Previous Observations	26
2.9.2. Mechanism responsible for stall delay	27
2.9.3. S809 aerofoil characteristics and polar extrapolation.....	31
2.9.4. 3D stall delay correction models	43
2.10 Inverse BEM.....	47
2.11 Implementation of computed $C_{L,3D}$ and $C_{D,3D}$ in BEM analysis for wind farm analysis.....	51
CHAPTER 3: WAKE AERODYNAMICS AND MODELLING	55
3.1. Atmospheric Boundary Layer Modelling	55
3.1.1. Introduction	55
3.1.2. Geostrophic wind.....	55
3.1.3. Atmospheric boundary layer (ABL).....	56
3.1.4. Thermal stratification	58
3.1.5. Modelling of neutral homogenous ABL.....	60
3.2. Wake Aerodynamics	61
3.2.1. Introduction	61
3.3. Wake Models	67
3.3.1. Challenges and difficulties in wake modelling.....	67
3.3.2. Near wake models	67
3.3.3. Far wake models.....	72
3.4. Turbulence Intensity Modelling.....	82
3.5. Hybrid models.....	86
CHAPTER 4: GOVERNING EQUATIONS AND NUMERICAL METHODS	87
4.1. Governing Equations.....	87
4.2. Reynolds Averaging	87
4.3. Turbulence Modelling (TM)	89
4.3.1. k- ω SST model.....	90
4.3.2. Standard k- ϵ model.....	94
4.3.3. Large Eddy Simulation (LES).....	96
4.4. Modelling Approaches of Rotating Machineries using FLUENT	100
4.4.1. Single Reference Frame (SRF)	100
4.4.2. Multiple Reference Frame (MRF)	101

4.4.3. Sliding Mesh (SM)	102
CHAPTER 5: CFD SIMULATION OF WIND TURBINE	103
5.1. NREL Phase VI Turbine	103
5.1.1. Experimental data	103
5.1.2. Aerodynamic Force Coefficients	106
5.2. CFD analysis of NREL Phase VI Turbine for the analysis of stall delay	107
5.2.1. Geometrical modelling	108
5.2.2. Computational Mesh	110
5.2.3. FLUENT Solver set up and Simulation	112
5.3. CFD simulations of NREL Phase VI Turbine in different terrains	113
5.3.1. Geometrical Modelling	113
5.3.2. Pre-processing of ABL simulation analysis	114
5.3.3. FLUENT setup for wind turbine simulation in ABL	116
CHAPTER 6: RESULTS AND DISCUSSION	121
6.1. CFD results of NREL Phase VI Turbine for uniform inflow	121
6.2. Computation of $C_{L,3D}$, $C_{D,3D}$ and AoA along the blade span by using Inverse BEM	137
6.3. Comparison of predicted $C_{L,3D}$ and $C_{D,3D}$ from inverse BEM with values computed along the blade span in BEM analysis without stall delay model	142
6.4. Comparison of predicted $C_{L,3D}$ and $C_{D,3D}$ from inverse BEM with values computed along the blade span in BEM analysis with stall delay model.	151
6.5. Improved BEM analysis for stall delay by implementing $C_{L,3D}$ and $C_{D,3D}$ computed from Inverse BEM analysis	155
6.6. Thrust, Torque and Power comparisons of NREL Phase VI Turbine (sequence S)	158
6.7. Analysis of wake characteristics of NREL Phase VI Turbine	161
6.7.1. Wake characteristics for different uniform inflow velocities	161
6.7.2. Wake characteristics under different ABL	178
CHAPTER 7: CONCLUSION AND FUTURE WORKS	196
7.1. Conclusions	196
7.2. Future works for improving indirect rotor modelling (Coupling between BEM-RANS)	197
7.3. Future works for improving wind farm analysis	200
7.4. Future works for improving turbulence intensity analysis in the wake	200
REFERENCES	202
CHAPTER 1	202
CHAPTER 2	202
CHAPTER 3	207
CHAPTER 4	214

CHAPTER 5	215
CHAPTER 6	215
CHAPTER 7	217
Appendix A: BEM MATLAB code (Buhl's Correction Factor and Lazafame and Messina's model of aerofoil characteristics extrapolation).....	219
APPENDIX B: Inverse BEM MATLAB code	223
APPENDIX C: Proposed BEM MATLAB code	254
APPENDIX D: UDF for Neutral ABL at inlet of the domain.....	259
APPENDIX E: NREL Phase VI blade geometry data	261
APPENDIX F: TURBINEinABL.jou (for $z_0=0.001\text{m}$)	262

LIST OF FIGURES

Figure 1.1: Flow over an aerofoil [9]	6
Figure 2.1: Control volume of actuator disk model [8].....	10
Figure 2.2: Wake rotation [8]	14
Figure 2.3: Annular stream tube control volume [8]	14
Figure 2.4: Blade element model [6].....	16
Figure 2.5: Blade element geometry, velocities and forces [6]	17
Figure 2.6: Flowchart for BEM analysis	23
Figure 2.7: Comparison of lift coefficient for rotating (RB) and non-rotating (NRB) blades at 30% blade span [37].....	27
Figure 2.8: Stall delay Phenomenon [44].....	28
Figure 2.9: Favourable pressure gradient due to the Coriolis force [45].....	28
Figure 2.10: Limiting streamlines on the suction side for wind speed 7m/s along with the pressure contour on the blade	29
Figure 2.11: Limiting streamlines on the suction side for wind speed 10m/s along with the pressure contour on the blade	29
Figure 2.12: Aerofoil characteristics of S809 aerofoil from various experiments [8]. (a) lift coefficients of S809 aerofoil (b) drag coefficients of S809 aerofoil	32
Figure 2.13: C_L extrapolation from 0° to 90° for S809 aerofoil using Viterna-Corrigan model [53]	33
Figure 2.14: C_D extrapolation from 0° to 90° for S809 aerofoil using Viterna-Corrigan model [53]	34
Figure 2.15: Power curves using Viterna-Corrigan model [53] from BEM MATLAB code (left) and Qblade (Right)	34
Figure 2.16: C_L , C_D vs AoA for a flat plate [54].....	35
Figure 2.17: C_L extrapolation from 0° to 90° for S809 aerofoil using Lanzafame and Messina method [10]	38
Figure 2.18: C_D extrapolation from 0° to 90° for S809 aerofoil using Lanzafame and Messina method [10]	38
Figure 2.19: Power curves using Lanzafame and Messina [10] proposal from BEM MATLAB code (left) and Qblade (Right)	39
Figure 2.20: C_L and C_D extrapolation from 0° to 90° for S809 aerofoil using Tangler and Kocurek proposal [54].....	41

Figure 2.21: Power curves using Tangler and Kocurek [54] proposal from BEM MATLAB code (left) and Qblade (Right)	41
Figure 2.22: Figures extracted from Tangler and Kocurek [54]. (a) Figure 6 (C_L and C_D versus wind speed) (b) Figure 9 (C_L and C_D versus Angle of Attack)	42
Figure 2.23: Streamline along the section 30% of blade span for wind speed 20 m/s.	43
Figure 2.24: Power curves predicted with different stall delay models	46
Figure 2.25: Comparison of classical BEM and Inverse BEM in an element. (a) classical BEM in an element (b) inverse BEM based on Guntur and Sørensen [60].....	48
Figure 2.26: Flowchart for improved inverse BEM	50
Figure 2.27: Aerodynamic force coefficients [61]	50
Figure 2.28: Angle of attack (left) and C_L (right) distributions along the blade span for wind speed 15m/s.....	52
Figure 2.29: Method for distribution of 3D aerofoil characteristics computed from pressure measurements.....	53
Figure 2.30: Comparison of classical BEM and proposed BEM. (a) classical BEM (b) proposed BEM.....	54
Figure 3.1: Atmospheric boundary layer and free atmosphere above earth's surface [5]	56
Figure 3.2: Variation of atmospheric boundary layer height for different terrains [6].....	57
Figure 3.3: Effect of friction with ABL height [8]	58
Figure 3.4: Vertical velocity profile for unstable, stable and neutral boundary layer [10]	60
Figure 3.5: Wind farm wake -Danish offshore wind farm "Horns Rev 1" [26].....	62
Figure 3.6: Cylindrical shear layer created by the tip vortices from a three-blade turbine [18]	64
Figure 3.7: Near wake region and far wake region behind the wind turbine [27]	65
Figure 3.8: Velocity profile of the wake along the horizontal plane at hub height [27]	65
Figure 3.9: Turbulence intensity profile of the wake along the horizontal plane at hub height [27]	65
Figure 3.10: Jensen wake model [82]	74
Figure 3.11: Flow behind the turbine based on Ainslie model [83]	80
Figure 4.1: An example of SRF model [7]	101
Figure 4.2: Mixing tank with single impeller [6]	102
Figure 4.3: Mesh motion in Sliding mesh approach [7]	102
Figure 5.1: NREL Phase VI 2-bladed turbine in the NASA Ames Wind Tunnel [1]	104
Figure 5.2: Location of pressure taps on the blade [1]. (a) pressure tap distribution on the blade span (b) Pressure taps distribution along the aerofoil section at the prescribed radial locations	106

Figure 5.3: Distribution of aerofoil profiles at various sections along the blade span. (a) distribution of aerofoils around the twist axis (b) distribution of aerofoils along the blade span	109
Figure 5.4: Isometric view of the blades	109
Figure 5.5: Computational domains for the CFD simulations. (a) front view (b) lateral view	110
Figure 5.6: 3D view of the domains set up for ABL analysis with dimensions	114
Figure 6.1: Pressure distributions for 7 m/s	124
Figure 6.2: Pressure distributions for 10 m/s	125
Figure 6.3: Pressure distributions for 15 m/s	126
Figure 6.4: Pressure distributions for 20 m/s	127
Figure 6.5: Pressure distributions for 25 m/s	128
Figure 6.6: Static pressure contours along with limiting streamlines on the pressure side (left) and suction side (right) for inlet wind speed 7m/s	129
Figure 6.7: Static pressure contours along with limiting streamlines on the pressure side (left) and suction side (right) for inlet wind speed 10m/s	129
Figure 6.8: Static pressure contours along with limiting streamlines on the pressure side (left) and suction side (right) for inlet wind speed 15m/s	130
Figure 6.9: Static pressure contours along with limiting streamlines on the pressure side (left) and suction side (right) for inlet wind speed 20m/s	130
Figure 6.10: Static pressure contours along with limiting streamlines on the pressure side (left) and suction side (right) for inlet wind speed 25m/s	131
Figure 6.11: Streamlines around different blade sections for inlet wind speed 7m/s	132
Figure 6.12: Streamlines around different blade sections for inlet wind speed 10m/s ...	133
Figure 6.13: Streamlines around different blade sections for inlet wind speed 15m/s ...	134
Figure 6.14: Streamlines around different blade sections for inlet wind speed 20m/s ...	135
Figure 6.15: Streamlines around different blade sections for inlet wind speed 25m/s ...	136
Figure 6.16: Comparison of extracted sectional $C_{L,3D}$ from different works with inverse BEM at five radial locations	138
Figure 6.17: Comparison of extracted sectional $C_{D,3D}$ from different works with inverse BEM at five radial locations	139
Figure 6.18: Computed $C_{L,3D}$ at 18 radial locations along the blade span using the inverse BEM method at different inlet wind speeds	140
Figure 6.19: Computed $C_{D,3D}$ at 18 radial locations along the blade span using the inverse BEM method at different inlet wind speeds	141

Figure 6.20: Angle of attack computed along the blade span using inverse BEM for different inlet wind speeds	142
Figure 6.21: Comparison of computed C_L from the inverse BEM and BEM analysis using Viterna-Corrigan model [7]	145
Figure 6.22: Comparison of computed C_D from the inverse BEM and BEM analysis using Viterna-Corrigan model [7]	146
Figure 6.23: Comparison of computed C_L from the inverse BEM and BEM analysis using Lanzafame and Messina model [8]	147
Figure 6.24: Comparison of computed C_D from the inverse BEM and BEM analysis using Lanzafame and Messina model [8]	148
Figure 6.25: Comparison of computed C_L from the inverse BEM and BEM analysis using Tangler and Kocurek model [9]	149
Figure 6.26: Comparison of computed C_D from the inverse BEM and BEM analysis using Tangler and Kocurek model [9]	150
Figure 6.27: Comparison of computed C_L from the inverse BEM and BEM analysis using different stall delay models	153
Figure 6.28: Comparison of computed C_D from the inverse BEM and BEM analysis using different stall delay models	154
Figure 6.29: Comparison of sectional C_L along the blade span from proposed BEM model with the values of CFD computed using inverse BEM	156
Figure 6.30: Comparison of sectional C_D along the blade span from proposed BEM model with the values of CFD computed using inverse BEM	157
Figure 6.31: Comparison of thrust calculated at different wind speeds by different methods	158
Figure 6.32: Comparison of torque values calculated at different wind speeds. (a) comparison of values from proposed BEM model and CFD analysis with experimental estimated aerodynamic torque (b) comparison of values from proposed BEM model and CFD analysis with experimental LSST	159
Figure 6.33: Comparison of predicted power from BEM and CFD analyses and measured power from experiment	161
Figure 6.34: Far wake velocity profiles comparisons for uniform inflow 7m/s	164
Figure 6.35: Far wake velocity profiles comparisons for uniform inflow 10m/s	165
Figure 6.36: Far wake velocity profiles comparisons for uniform inflow 15m/s	166
Figure 6.37: Far wake velocity profiles comparisons for uniform inflow 20m/s	167
Figure 6.38: Far wake velocity profiles comparisons for uniform inflow 25m/s	168

Figure 6.39: Contours of U/U_{ref} in horizontal sectional plane taken at hub height.....	169
Figure 6.40: Turbulence intensity profiles at inlet and downstream distances of rotor for uniform inflow 7m/s	172
Figure 6.41: Turbulence intensity profiles at inlet and downstream distances of rotor for uniform inflow 10 m/s.....	173
Figure 6.42: Turbulence intensity profiles at inlet and downstream distances of rotor for uniform inflow 15 m/s.....	174
Figure 6.43: Turbulence intensity profiles at inlet and downstream distances of rotor for uniform inflow 20 m/s.....	175
Figure 6.44: Turbulence intensity profiles at inlet and downstream distances of rotor for uniform inflow 25m/s	176
Figure 6.45: Comparison of averaged turbulence intensity with in wake diameter and different existing models.....	177
Figure 6.46: Inlet velocity profiles for different ABL.....	178
Figure 6.47: Pressure comparisons along the blade span at selected radial locations between experiment results with uniform inlet flow of 10 m/s and CFD with uniform inflow and ABL with different roughness lengths.....	180
Figure 6.48: Horizontal lateral profiles of normalized streamwise velocity comparisons at different downstream distances under ABL with different roughness lengths and uniform inflow	182
Figure 6.49: Contours of U/U_{ref} along the horizontal plane under ABL with different roughness lengths	183
Figure 6.50: Vertical profiles of normalized streamwise velocity comparisons along z axis at different downstream distances under ABL with different roughness lengths and uniform inflow	184
Figure 6.51: Contour of U/U_{ref} along the vertical midplane under ABL with different roughness lengths	185
Figure 6.52: Horizontal lateral profiles of turbulence intensity at inlet and downstream distances of the rotor and roughness length of ABL z_0 is 0.001m	187
Figure 6.53: Horizontal lateral profiles of turbulence intensity at inlet and downstream distances of the rotor and roughness length of ABL z_0 is 0.01m	188
Figure 6.54: Horizontal lateral profiles of turbulence intensity at inlet and downstream distances of the rotor and roughness length of ABL z_0 is 0.1m	189
Figure 6.55: Horizontal lateral profiles of turbulence intensity at inlet and downstream distances of the rotor and roughness length of ABL z_0 is 0.7m	190

Figure 6.56: Vertical profiles of turbulence intensity along z axis at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.001m	192
Figure 6.57: Vertical profiles of turbulence intensity along z axis at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.01m	193
Figure 6.58: Vertical profiles of turbulence intensity along z axis at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.1m	194
Figure 6.59: Vertical profiles of turbulence intensity along z axis at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.7m	195

LIST OF TABLES

Table 2.1: Different C_L at 45° and C_D at 90° predicted for different Reynolds numbers from the reference [8]	37
Table 2.2: Different stall delay models	45
Table 3.1: Important types of terrains and their corresponding roughness length [7]	58
Table 3.2: Summary of input parameters for different kinematic models	79
Table 3.3: Summary of input parameters for different wake turbulence intensity models	86
Table 4.1: Model constants for standard k- ϵ model	96
Table 4.2: RANS Vs LES	100
Table 5.1: Different cases of the NREL Phase VI experiment [1]	104
Table 5.2: Rotor description of NREL Phase VI Turbine [1]	105
Table 5.3: Maximum y^+ values on blade surface for the selected wind speeds	112
Table 5.4: Parameters for creating outer domain mesh of the adjacent layer of the ground	116
Table 5.5: Values of friction velocity for different terrains	117
Table 5.6: Values applied at top boundary as inlet for different terrains	119

ACRONYMS

ABL	Atmospheric Boundary Layer
AD	Actuator Disk
AL	Actuator Line
AoA	Angle of Attack
AR	Aspect Ratio
AS	Actuator Surface
BEM	Blade Element Momentum
CFD	Computational Fluid Dynamics
CSU	Colorado State University
DES	Detached Eddy Simulation
DNS	Direct Numerical Simulation
DUT	Delft University of Technology
DWW	Dutch Wind Workshops
ECN	Energy Research Center of the Netherlands
EWEC	European Wind Energy Conference
EWTS	European Wind Turbine Standard II
GUI	Graphical User Interface
HAWT	Horizontal Axis Wind Turbine
HPC	High Performance Computing
LES	Large Eddy Simulation
LSST	Low Speed Shaft Torque
LSWT	Lifting Surface Wake Theory
MATLAB	MATrix LABoratory
MRF	Multiple Reference Frame
NACA	National Advisory Committee for Aeronautics
NASA	National Aeronautics and Space Administration

NRB	Non-Rotating Blade
NREL	National Renewable Energy Laboratory
NSF	National Science Foundation
NTU	Nanyang Technological University
OSU	Ohio State University
RANS	Reynolds Averaged Navier-Stokes
RB	Rotating Blade
RHS	Right Hand Side
RPM	Revolutions per minute
SGS	Sub Grid Scale
SM	Sliding Mesh
SRF	Single Reference Frame
SST	Shear Stress Transport
TM	Turbulence Modelling
TSR	Tip Speed Ratio
UAE	Unsteady Aerodynamics Experiment
UDF	User Defined Function
URF	Under Relaxation Factor
VAWT	Vertical Axis Wind Turbine
WAsP	Wind Atlas Analysis and Application Program

NOMENCLATURE

a	Axial induction factor	
a_c	Critical axial induction factor	
a_1	Speed of the sound	(m/s)
a'	Angular induction factor	
A	Area	m^2
A_{wake}	Wake area	m^2
A_1	Constant	
b	Wake width	m
B	Number of blades	
c	Chord of the blade section (aerofoil)	m
c_1	Non-dimensional mixing length	
C_D	Coefficient of drag	
C_{DP}	Pressure drag coefficient	
$C_{D,s}$	Coefficient of drag at stall angle of attack	
$C_{D,2D}$	Coefficient of drag for 2D flow	
$C_{D,3D}$	Sectional drag coefficient corrected for 3D effect (stall delay)	
C_M	Coefficient of moment	
C_{NOR}	Normal force coefficient	
C_L	Coefficient of lift	
$C_{L,s}$	Coefficient of lift at stall angle of attack	
$C_{L,2D}$	Coefficient of lift for 2D flow	
$C_{L,3D}$	Sectional lift coefficient corrected for 3D effect (stall delay)	
C_p	Coefficient of pressure	
$C_{p,u}$	Coefficient of pressure along suction side of aerofoil	
$C_{p,l}$	Coefficient of pressure along pressure side of aerofoil	
C_P	Power coefficient	
C_s	Roughness constant	
C_{s1}	Smagorinsky constant	
C_T, C_{THRUST}	Thrust coefficient	

C_{TAN}	Tangential force coefficient	
C_{TORQUE}	Torque force coefficient	
C_{μ}	Constant	
$C_{1\varepsilon}$	Constant used in k- ε model	
$C_{2\varepsilon}$	Constant used in k- ε model	
C_2	Pressure jump coefficient	1/m
d	Distance to the closest wall	m
D	Diameter	m
D_{eff}	Effective rotor diameter	m
D_W	Wake Diameter	m
D_{ω}	Cross diffusion term	kg/(m ² s ²)
D_{ω}^{+}	Positive portion of the cross diffusion term	kg/(m ² s ²)
D_1	Fluid domain	
Drag	Drag force of 2D aerofoil (unit width)	N/m
f	Elliptical relaxation factor	
f_{CD}	Function for drag coefficient 3D correction	
f_{CL}	Function for lift coefficient 3D correction	
f_{hub}	Factor used to determine F_{hub} and is defined in Equation (2.63)	
f_{tip}	Factor used to determine F_{tip} and is defined in Equation (2.54)	
f_n	polynomial coefficient	
f'	Body force per unit mass	N/kg
$\vec{f}$	force vector	N
F	Correction factor	
$F_{1,2}$	Blending function constant	
$F_{\text{centrifugal}}$	Centrifugal force	N
$\vec{F}_{\text{Coriolis}}$	Coriolis force vector	N
F_{drag}	Drag force	N
F_{hub}	Hub loss correction factor	
F_{lift}	Lift force	N
F_{tip}	Tip loss correction factor	
g_i	Gravitational vector's component in i^{th} direction	(m/s ²)

G	Filter function	
G_b	Turbulence kinetic energy generation by buoyancy	$\text{kg}/(\text{ms}^2)$
$G_k, \tilde{G}_k$	Turbulence kinetic energy generation by mean velocity gradients ($\tilde{G}_k = \min(G_k, 10 \rho \beta^* k \omega)$)	$\text{kg}/(\text{ms}^2)$
G_ω	Generation of ω	$\text{kg}/(\text{m}^3\text{s}^2)$
I	Turbulence intensity	
I_a	Ambient turbulence intensity	
I_w, I_{wake}	Wake turbulence intensity	
I_+	Added turbulence intensity	
I_{++}	Added turbulence above the hub height due to additional surface roughness from the turbines in the farm	
k	Turbulence kinetic energy	m^2/s^2
k_s	Roughness Height	m
k_{spera}	Parameter defined by Equation (2.61) which is used in Glauert correction Model for axial induction factor	
K	von Karman constant	
K_L	Parameter defined by Equation (2.73) which is used in Viterna and Corrigan model	
K_D	Parameter defined by Equation (2.74) which is used in Viterna and Corrigan model	
K_w	Wake decay constant	
k_1	Constant	
L	Torque force	N
Le	Characteristic length	m
L_s	Mixing length of the subgrid-scales	m
Lift	Lift force of 2D aerofoil (per unit width)	N/m
m	Mass	kg
m_1	Parameter which is used to determine near wake distance and is defined by Equation (3.45)	
M_t	Turbulent Mach number	
$\dot{m}$	Mass flow rate	kg/s
Moment	Moment of the 2D aerofoil (per unit width)	N
Δm	Disk thickness	m

p	Pressure	Pa
p_{dynamic}	Dynamic pressure	Pa
$\bar{p}$	Mean component of the pressure term	Pa
p'	Fluctuation component of the pressure term	Pa
p''	Pressure drop across rotor plane	Pa
$\hat{p}$	Filtered pressure	Pa
p_w	Parameter which is defined by Equation (3.28)	
P	Power	W
Pr_t	Turbulent Prandtl number for energy	
Q	Torque	Nm
r	Blade local radius	m
r_o	Parameter which is used to determine near wake distance and is defined by Equation (3.46)	
r_w	Wake local radius	m
R	Radius of the rotor or disk	m
$\bar{R}$	Gas constant ($8.3145 \text{ J mol}^{-1} \text{ K}^{-1}$)	$\text{J mol}^{-1} \text{ K}^{-1}$
R_{nb}	Empirical relation for $R_{9.5}$ which is defined by Equation (3.21)	m
$R_{9.5}$	Radius of the wake at 9.5 times rotor diameter behind the turbine	m
Re	Reynolds number	
s	Relative distance downstream behind the rotor (normalized by the rotor diameter)	
s_1	Inter-turbine spacings (normalized by the rotor diameter) within a row of a wind farm	
s_2	Inter-turbine spacings (normalized by the rotor diameter) between the rows of a wind farm	
S	Strain rate magnitude	
S_{ij}	Mean rate of strain tensor	s^{-1}
$\hat{S}_{ij}$	Rate of strain tensor for the resolved scale	s^{-1}
S_ε	User defined source term for turbulent dissipation rate	$\text{kg}/(\text{ms})$
ε		
S_k	User defined source term for turbulence kinetic energy	$\text{kg}/(\text{ms}^2)$
k		

S_ω	User defined source term for specific dissipation rate ω	kg/(m ³ s ²)
t	Time	s
T	Thrust force	N
T'	Time interval	s
T _e	Absolute temperature	K
u,U	Velocity magnitude	m/s
u _c	Velocity at centre line of wake	m/s
U _{ref}	Reference velocity above the ground at z _{ref}	m/s
U _w	Wake velocity	m/s
U _{w0}	Wake velocity at zero distance downstream of the rotor	m/s
$\bar{u}$	Mean wind velocity	m/s
u _*	Friction velocity	m/s
$\vec{u}$	Velocity vector	m/s
u'	Fluctuating component of velocity	m/s
$\hat{u}$	Filtered velocity	m/s
v	Velocity in radial direction	m/s
v ²	Velocity scale in v ² -f turbulence model	
V	Volume	m ³
$\vec{V}$	Velocity vector	m/s
Vel	Resultant velocity vector	m/s
x	Distance from the origin along the positive x axis (used also to represent downstream distance provided origin is at the rotor plane)	m
x _c	Normalized distance by the blade chord along the chord line	
x _n	Distance of near wake region	m
x _o	Rotor position of applied coordinate system	m
y	Distance from the origin along the positive y axis (used also to represent downstream distance provided origin is at the rotor plane)	m
y ₁	Coordinate normal to the wall or no slip boundary	m
y _c	Normalized distance by the blade chord orthogonal to the chord line	

y^+	Dimensionless distance to the wall	
Δy	Distance of the first node from the wall	(m)
Y_k	Term for dissipation of k due to turbulence	kg/(ms ²)
Y_M	Dilation diffusion term	kg/(ms ²)
Y_ω	Term for dissipation of ω due to turbulence	kg/(m ³ s ²)
z	Height above the ground	m
z_{hub}	Hub height	m
z_o	Roughness length	m
z_{ref}	Reference height above the ground	m

Greek Symbols

α	Angle of attack	rad
α_o	Angle of attack at zero lift	rad
α_1	Constant	
$\alpha_{porousmedia}$	Permeability of porous media	m ²
$\alpha_2, \alpha_\infty, \alpha_{\infty,1},$ $\alpha_{\infty,2}, \alpha_i^*, \alpha_\infty^*$	Coefficients used in k- ω SST model	
α_s	Stall angle of attack	rad
β	Twist angle (blade pitch angle+local twist angle)	rad
β_1	Wake expansion parameter	
β_2	Thermal expansion coefficient	(1/K)
$\beta_i, \beta_i^*, \beta_i^*$	Coefficients used in k- ω SST model	
δ	Kronecker delta	
ε	Turbulent dissipation rate	m ² /s ³
$\varepsilon(x)$	Eddy viscosity	m ² /s
ϕ	Angle between the relative velocity and plane of rotation	rad
ϕ_1	General fluid variable (pressure or velocity)	
ϕ_{F1}, ϕ_{F2}	Coefficients used in k- ω SST model	
Φ	Velocity potential	m ² /s
γ	Ratio of the specific heats	
Γ_k	Effective diffusivity of k	Ns/m ²

Γ_{ω}	Effective diffusivity of ω	Ns/m ²
λ	Tip speed ratio	
λ_r	Local tip speed ratio	
Λ	Modified tip speed ratio	
μ	Dynamic viscosity	Ns/m ²
μ_t	Turbulent (eddy) viscosity	Ns/m ²
μ_{SGS}	Sub grid scale (eddy) viscosity	Ns/m ²
ν	Kinematic viscosity	m ² /s
ν_t	Kinematic turbulent (eddy) viscosity	m ² /s
ρ	Air density	kg/m ³
σ	Solidity	
σ	Turbulent Prandtl number	
σ_{ε}	Turbulent Prandtl number of ε	
σ_k	Turbulent Prandtl number of k	
σ_u	Standard deviation of wind speed variations about the mean wind speed	m/s
σ_{ω}	Turbulent Prandtl number of ω	
τ	Stress	N/m ²
τ'_{ij}	Reynolds stress tensor	N/m ²
τ^s_{ij}	Subgrid-scale stress	N/m ²
τ^s_{kk}	Isotropic part of the subgrid-scale stresses	N/m ²
τ_W	Wall shear stress	N/m ²
ω	Angular velocity imparted to the wake	rad/s
ω	Specific dissipation rate of turbulence kinetic energy, k	1/s
Ω	Angular velocity of rotor	rad/s
$\vec{\Omega}$	Angular velocity vector	rad/s

Superscriptions

(-)	Time averaged (mean) value
(^)	Filtered value
(')	Fluctuated value from mean

Subscriptions

2D	Two-Dimensional
3D	Three-Dimensional
a, ambient	Ambient
after	After the rotor
before	Before the rotor
c	Critical
drag	Drag of aerofoil
dynamic	Dynamic
eff	Effective
lift	Lift of aerofoil
hub	Hub
max	Maximum
mechanical	Mechanical
NOR	Normal
ref	Reference
rel	Relative
rotor	Rotor or Turbine or Disk
s	Stall
sheargenerated	Shear generated
t	Turbulence
tip	Tip of the blade
TAN	Tangential
W, wake	Wake
i,j,k	Covariant components
∞, o	Freestream region

Other Symbols

max (x,y)	Choose the maximum term between x and y
min (x,y)	Choose the minimum term between x and y

SUMMARY

Steadily increasing energy consumption, fluctuating fuel costs and concerns about global climate changes have led to the research and evaluation of alternative renewable energies. The wind turbine is among the promising alternative energy sources that have recently gained more attention. In this work, the challenging aspects involved in modelling the rotor aerodynamics and wakes behind the wind turbines are studied.

This report covers the literature review of Blade Element Momentum (BEM) analysis of wind turbine, limitations of BEM method, effect of stall delay, wake aerodynamics, atmospheric boundary layer and its effects on the wake characteristics and previous different wake models of near and far wake regions. In this project, experimental data of NREL Phase VI Turbine (sequence S) were used to corroborate the results. There are two main objectives of this project. The first objective is to improve the BEM analysis to account for the three-dimensional (3D) effects due to the rotation of the turbine. The second objective is to contemplate on the effects of atmospheric boundary layers (ABL) on the wake characteristics. These two objectives act as a roadway to enhance coupled BEM-CFD analysis, which is left for the future works. In the coupled BEM-CFD analysis, the BEM method will be applied to calculate the aerodynamic forces of the aerofoil sections along the blade span. The main drawback of BEM analysis is the use of the two-dimensional (2D) aerofoil characteristics (C_L and C_D) which considers only the axial flow but not radial or spanwise flow along the blade span. This leads to a considerable difference in the lift coefficients between the rotating and non rotating blades, especially at inboard sections of the blade. This 3D phenomenon is called stall delay. The current study includes different proposals for the extrapolation of

2D aerofoil characteristics of the S809 aerofoil and their implementation in BEM analysis and comparison of power predicted with experimental results. Also, four existing stall delay correction models in BEM analysis are examined. In general, these models result in over-prediction of power, especially at high inflow wind speeds. An improved inverse BEM method is developed to compute 3D aerofoil characteristics at different radial locations along the blade span. In addition to five radial locations as described in the NREL/NASA Ames test analysis, 13 additional radial locations are considered for a better understanding of stall delay. A new BEM model with the local radius effect of aerofoil characteristics (other than as a function of Reynolds number and angle of attack only) is proposed. Implementation of the new model showed a good agreement with aerofoil characteristics distribution along the blade span with the 3D aerofoil characteristics computed from the CFD analyses using inverse BEM method. MATLAB code was developed for both BEM and Inverse BEM analyses.

Most important in the wind farm analysis is the effect of the atmospheric boundary layer since the turbulence properties of the atmosphere affect the wake characteristics. In this work, the NREL Phase VI Turbine is virtually placed in different atmospheric boundary layers from open sea to city/forest. Simulations are performed with direct rotor modelling using sliding mesh analysis. Since the experiment results of wake characteristics of the NREL Phase VI Turbine was not available, different empirical models are used for comparison. It was evident that wake recovers at a faster rate as the roughness length of the ground increases. Also, it was noted that the turbulence intensity of the wake varies both laterally and vertically, but existing analytical wake turbulence intensity models provide only an averaged value.

The possible methods for indirect rotor modelling like Actuator Disk, Actuator Line and Actuator Surface methods that lead to coupled BEM-CFD analysis of wind turbine is discussed in brief. The possible ways of improving these indirect rotor models by using the aerodynamic forces computed from improved BEM analysis and wake aerodynamics are provided for the future works.

CHAPTER 1: INTRODUCTION

1.1. Energy need in the world

Energy is now widely regarded as an essential marker of social and economic development. If a nation is prosperous, it is certain to have a high rate of energy consumption. Energy usage has risen exponentially in both developed and developing countries. In fact, studies show a recent increase of 5.6 percent in global energy use, which is the highest increase since 1973 [1].

The world's energy needs are met from different sources. The U.S. Energy Information Administration states that fossil fuels account for about 86% of global energy production [2], while the remaining is accounted for by nuclear energy and renewable sources such as biomass, hydropower, wind, tidal and solar. If the use of fossil fuels continues at the present rate, it would increase atmospheric carbon dioxide by 10.65 billion tones per year [3]. A remedy is to use renewable energy sources instead, which are progressively maturing enough to substitute fossil fuels.

It is believed that renewable sources would get more focus if their benefits were given full credit. With today's technology, wind energy is favored as an alternative, aside from being clean, it is abundant and widely spread out. Additionally, wind energy produces no greenhouse gas emissions. Therefore, large scale wind turbines for public use have started emerging since the last century [4].

1.2. Energy resources in the world

To fulfill the energy needs projected for the future, it is crucial to assess the availability of conventional energy as well as renewable energy.

1.2.1. Conventional energy

The conventional energies are fossil fuels and nuclear resources. Fossil fuel includes coal, petroleum and natural gas and it has been the fuel of choice for electricity generation in the recent past.

1.2.2. Renewable energy

Renewable energy has a pervasive influence on human life. The question is the sustainable and efficient utilization of renewable energy sources. Sources with high global penetration include hydro, wind, tidal, ocean and solar, while sources like geothermal power and biogas have a smaller global influence.

One renewable technology that has seen marked improvement in recent years, both in cost and efficiency, is that of wind energy [5]. Especially where wind resources are abundant, the energy production can be substantial. Wind energy is thus significantly contributing to the worldwide energy production.

1.3. Historical development of wind turbines

Mankind made efforts to use the wind as energy for at least 3000 years back as sails to ship and boats [6]. It was then applied to water pumps and grain grinding mills. Scholars differ about the origins of wind use, but the Dutch are known to have been among the first to use the mills. They enhanced the design of both the mills and the rotors used in them [7]. Making use of wind has since evolved from rudimentary devices to advanced machinery, undergoing different stages in the process.

In the late nineteenth century, wind electric generators came to the fore. The Danish designed and used wind turbines in 1900 that set the trend for modern ones [7]. The viability of large scale wind generation was demonstrated with the Smith-Putnam wind turbine. Other designs, such as the Darrieus turbine and Savonius rotor, were also present in the early twentieth century [7]. The interest in wind energy rose significantly after the oil crisis in the 1970's when more countries started turning to renewable sources of energy [7].

In 1972, the National Science Foundation (NSF) and National Aeronautics and Space Administration (NASA) helped found the Federal Wind Energy Program in the United States. This program commissioned the Lewis Research Center to develop and deploy large wind turbines. Additionally, the center conducted large-scale research on enhancing wind energy technology. Other countries such as the United Kingdom (UK), Germany and

Sweden also invested in wind energy research. In the process, large wind turbines were seen to have a number of disadvantages. As an alternative, a high number of small wind turbines that would be easy to repair and modify started to be installed. The search for cost-effective and simple machines lead to the design of the Savonius turbine in Finland [8]. Recently, wind energy is being increasingly generated in wind farms offshore.

1.4. Recent development of wind turbines

The era of modernized wind turbines garnered strength largely in response to the many issues with the use of fossil fuels as sources of energy. Aside from being limited, fossil fuels cause alarming carbon emissions. Thus, in the 20th century, the integration of various disciplines such as aerodynamics, computer science, material science and power electronics significantly enhanced wind turbines. Aerodynamic designs adapted for aerospace industry are used in wind turbines. The design, analysis, monitoring and control aspects are the domain of computer science. New alloys used in the metal parts of turbines are the fruit of material science. The connection of the turbine to the electrical network and variable speed operations are the domain of power electronics. Additionally, large numbers of sensors have to be tested and data has to be analyzed to assess the performance of new turbines.

Over the last 25 years, wind turbines have increased in both cost-effectiveness and reliability. Turbines used in remote communities can be made quite productive. There are promising opportunities in offshore locations. To sustain wind turbines for long periods and increase their productivity, it is important to implement advanced engineering methods in the many stages of their development. The specialized application of the turbines needs to be given due consideration too [4].

1.5. Aerofoil used in wind turbine blades

1.5.1. Preamble

Wind turbine blades have aerofoil cross section. Geometry and the aerodynamic properties of the aerofoil determine the power produced by the wind turbine. It is necessary to predict the important characteristics of the aerofoil like lift and drag. The predicted lift and drag aerofoil characteristics are used to calculate the wind turbine blade properties. The most famous aerofoils are produced by National Advisory Committee for Aeronautics (NACA). There are many aerofoil families developed by research institutes such as Risø National University in Denmark, Delft University in Netherland and National Renewable Energy Laboratory (NREL) in the US for wind turbine applications.

The aerodynamics and stall condition of aerofoil are revised briefly in the following section.

1.5.2. Aerodynamics of Aerofoil

As the flow past the aerofoil, the velocity of the air on the upper (suction) surface increases and pressure decreases, in contrary the velocity of the air decreases in the lower (pressure) surface and the pressure increases. Also, friction between the air and the surfaces reduce the velocity of the air. The pressure difference between the lower (pressure) surface and upper (suction) surface of the aerofoil and skin friction drag create a net force and moment. The net force acting on the aerofoil can be resolved into two components. The lift force is perpendicular to the air direction and the drag force is along the air direction. The aerodynamic centre of an aerofoil is usually close to 25% of the chord behind the leading edge of the aerofoil and corresponding pitching moment at this point is given as in Equation (1.3).

$$\text{Lift} = C_L \left(\frac{1}{2} \right) \rho U^2 c \quad (1.1)$$

$$\text{Drag} = C_D \left(\frac{1}{2} \right) \rho U^2 c \quad (1.2)$$

$$\text{Moment} = C_M \left(\frac{1}{2} \right) \rho U^2 c^2 \quad (1.3)$$

where,

C_L =Coefficient of lift

C_D =Coefficient of drag

C_M =Coefficient of moment

c is the aerofoil chord, i.e. the distance between the leading edge and trailing edge, measured along the chord line of the aerofoil.

C_L and C_D values will not be the same for all flow conditions. There are factors like velocity, pressure, temperature, viscosity and length that influence the values of C_L and C_D . These factors are represented by a non-dimensional number called Reynolds number.

$$Re = \frac{U * Le}{\nu} = \frac{\text{Inertial Force}}{\text{Viscous Force}} \quad (1.4)$$

where,

U = Reference velocity (freestream velocity) (m/s)

Le = Characteristic Length (chord) (m)

ν = Kinematic viscosity (m^2/s)

Aerofoils designed for high Reynolds number application (Aeroplanes) cannot be used for low speed applications (wind turbine except huge turbines). Therefore, it is necessary to select suitable aerofoil profile depending on the application.

1.5.2.1. Stall Condition

The value of C_L increases with an increase in the angle of attack. The rate of increase depends on the aerofoil design. The progressive increase in the angle of attack causes the boundary layer on the upper surface to detach from the blade surface resulting in the formation of vortices and flow separation occurs due to the adverse pressure gradient developed on the suction surface as shown in Figure 1.1. The aerofoil is said to be stalled as the lift decreases drastically. At this point, the critical angle of attack is reached. It is usually in the range 8° to 20° .

Figure 1.1: Flow over an airfoil [9]

At the time of operation of wind turbines, flow locally might not align with the designed inlet blade angle (incidence angle), thus the blade at designed angle of attack due to various situations which can result in stall at particular cross section causing the aerodynamic losses.

1.6. Aim of this study

- To analyze the cause and effect of stall delay which occurs due to the rotation of the wind turbine blades. Unlike as described above, due to the rotational effect occurrence of stall is delayed in wind turbine blade. This is a serious issue as existing models do not account accurately for stall delay.
- To compute 3D sectional airfoil characteristics from the rotor CFD analysis using the inverse BEM methodology. The conventional BEM method uses 2D airfoil characteristics (described in Section 1.5.2.) which lead to under-prediction of BEM with experimental analysis. A modified BEM model will be proposed by using the computed 3D airfoil characteristics from the inverse BEM method.
- To investigate the effect of different atmospheric boundary layers on the wake characteristics of the NREL Phase VI Turbine.
- To provide methodologies for improving the currently existing indirect rotor models like Actuator Disk, Actuator Line and Actuator Surface models.

1.7. Original contributions of the present study

- Improved inverse BEM method to compute 3D aerofoil characteristics from the CFD analysis of wind turbine rotor.
- The new method is proposed for improving BEM analysis to account for stall delay.
- Experimental analysis of the NREL Phase VI Turbine was performed previously in the wind tunnel and the experimental results do not account for wake characteristics. In the present work, extensive research was done on both rotor and wake aerodynamics of the NREL Phase VI Turbine by virtually placing it under different atmospheric boundary layers.

1.8. Limitations of the present study

- The limitations of conventional BEM theory are discussed in Section 2.8: Unsteady Aerodynamics and 3D Aspects of Chapter 2. The prime limitation of BEM in accounting for stall delay effect was taken for consideration and improved BEM method is proposed only to account for stall delay effect. Other limitations discussed in Chapter 2 are left for the future works.
- Two bladed NREL Phase VI was only used for analysis as precise experimental results were available only for it. But in real cases, wind turbines are usually three bladed.
- Since experimental results are not available for wake characteristics of the NREL Phase VI Turbine, only existing models are used for validation. Drawbacks of existing models are described in Chapter 6.

1.9. Publications based on current work

1.9.1. Journals

1. Ijaz Fazil Syed Ahmed Kabir, E.Y.K. Ng, Insight into Stall delay and Computation of 3D NREL Phase VI Turbine Aerofoil Characteristics using Inverse BEM and its

- Implementation in BEM Analysis to account for Stall Delay, *Energy*, 120 (2017): 518-536.
2. Ijaz Fazil Syed Ahmed Kabir, E.Y.K. Ng, Effect of the atmospheric boundary layer on wake characteristics of NREL Phase VI Wind Turbine, *Renewable Energy*. (Under review in *Renewable Energy*).

1.10. Outline of this thesis report

- In *Chapter 1*, energy demand in the world, historical development and modern growth of wind turbines, aerofoil used in wind turbines are presented. Also aim, contributions, limitations and publications are also included.
- In *Chapter 2*, extensive descriptions of analytical rotor modelling using BEM theory are covered. Introduction to stall delay, issues in S809 aerofoil characteristics extrapolation, drawbacks of existing stall delay models, a method to compute 3D aerofoil characteristics are explained in detail. In addition, a new approach for BEM is proposed to account for stall delay.
- In *Chapter 3*, introduction to the atmospheric boundary layer, wake aerodynamics, different near wake and far wake models are explained. Models for computing turbulence intensity in the wake are also included.
- In *Chapter 4*, introduction to turbulence modelling used in this work is presented.
- In *Chapter 5*, set up and CFD modelling of the NREL Phase VI Turbine for performing inverse BEM analysis and under different atmospheric boundary layer are described.
- In *Chapter 6*, results and discussion for both the rotor and wake aerodynamics based on previous chapters are discussed.
- In *Chapter 7*, a summary of the present work and possible mode for improving indirect rotor modelling using the present results in the future are proposed.

CHAPTER 2: AERODYNAMICS OF WIND TURBINES

2.1. Introduction

A wind turbine is a device that is used to extract kinetic energy from the wind and convert it into the mechanical energy to produce power. The power production depends primarily on the interaction between the wind and the rotor. Aerodynamic forces generated by the wind determine the major aspects like loads and power output.

This chapter focuses primarily on the steady-state aerodynamics and deals with the classical analytical approach [1-6] to analyze the performance of the horizontal axis wind turbines (HAWT). This approach mainly deals with the energy extraction process without any specific turbine design. Actuator Disc model is employed to calculate the power output of an ideal HAWT and the thrust on the turbine blades. A model called Blade Element Momentum theory (BEM) includes a combination of momentum theory and blade element theory. Performance parameters of an ideal HAWT under steady state operating conditions can be predicted using BEM theory. BEM theory is also used for the optimization of the blade shape. BEM theory combines two theories to formulate the governing equations for the aerodynamic design and the power prediction. Blade element theory determines the aerodynamic forces created by the lift and the drag coefficients at different sections along the blade span and the momentum theory examines the momentum balance on a rotating annular stream tube.

2.2. Actuator Disk Model

The actuator disk model [7] does not include any turbine design. It is only based on the energy extraction process. This is the simplest model in which turbine is replaced by a disk through which the wind flows and there is a pressure drop from p_{before} to p_{after} . This model predicts the overall efficiencies of the turbine.

Assumptions made in this model include:

- Steady state homogeneous incompressible flow

- No frictional drag
- Continuity of velocity through the disk
- Constant pressure increment over the disk
- Infinite number of blades
- A non-rotating wake
- One-dimensional flow

This analysis assumes the control volume around the disk as shown in Figure 2.1. The boundaries of the control volume are the walls of the stream tube and two cross sections. Four stations in the control volume are needed to be considered.

Figure 2.1: Control volume of actuator disk model [8]

- They are
0. Freestream region (∞)
 1. Region just before blade (before)
 2. Region just behind the blade (after) and
 3. Far wake region (wake)

$$p_{\infty} = p_{\text{wake}} = p_o \quad (2.1)$$

The mass flow is constant throughout the control volume

$$\rho A_{\infty} U_{\infty} = \rho A_{\text{rotor}} U_{\text{rotor}} = \rho A_{\text{wake}} U_{\text{wake}} \quad (2.2)$$

As stated in the assumption, there is continuity of velocity through the disk. Also, the area just before the blade and just after the blade will be same as the area of the rotor disk, hence

$$U_{\text{before}} = U_{\text{after}} = U_{\text{rotor}} \quad (2.3)$$

Since the flow is steady mass flow rate is

$$\dot{m} = \rho A U_{\text{rotor}} \quad (2.4)$$

By applying the conservation of linear momentum on both sides of disk gives thrust as

$$T = \dot{m}(U_{\infty} - U_{\text{wake}}) \quad (2.5)$$

By applying Bernoulli equation between Sections 2 and 3 and then between the Sections 0 and 1, the expressions (2.6) and (2.7) can be obtained respectively.

$$p_{\text{after}} + \left(\frac{1}{2}\right)\rho U_{\text{rotor}}^2 = p_o + \left(\frac{1}{2}\right)\rho U_{\text{wake}}^2 \quad (2.6)$$

$$p_o + \left(\frac{1}{2}\right)\rho U_{\infty}^2 = p_{\text{before}} + \left(\frac{1}{2}\right)\rho U_{\text{rotor}}^2 \quad (2.7)$$

From Equations (2.6) and (2.7), $p'' = (p_{\text{before}} - p_{\text{after}})$ can be expressed as

$$p'' = \left(\frac{1}{2}\right)\rho(U_{\infty}^2 - U_{\text{wake}}^2) \quad (2.8)$$

Thrust is given by the sum of the forces on both sides of the rotor

$$T = A p'' \quad (2.9)$$

When Equation (2.8) is substituted into Equation (2.9), Thrust can be expressed as

$$T = \left(\frac{1}{2}\right)A\rho(U_{\infty}^2 - U_{\text{wake}}^2) \quad (2.10)$$

From the Equations (2.4), (2.5) and (2.10) velocity through the rotor is given by

$$U_{\text{rotor}} = \left(\frac{U_{\infty} + U_{\text{wake}}}{2}\right) \quad (2.11)$$

By introducing a factor called axial induction factor (a) as follows

$$a = \frac{U_{\infty} - U_{\text{rotor}}}{U_{\infty}} \quad (2.12)$$

On rearranging

$$U_{\text{rotor}} = U_{\infty}(1 - a) \quad (2.13)$$

$$U_{\text{wake}} = U_{\infty}(1 - 2a) \quad (2.14)$$

The power output is given by

$$P = T U_{\text{rotor}} \quad (2.15)$$

By substituting Equation (2.10) into Equation (2.15), the following equation of power can be obtained:

$$P = \left(\frac{1}{2} \right) \rho A (U_{\infty}^2 - U_{\text{wake}}^2) U_{\text{rotor}} \quad (2.16)$$

And finally, by substituting Equations (2.13) and (2.14) into Equation (2.16), power can be expressed as:

$$P = 2 \rho A a (1 - a)^2 U_{\infty}^3 \quad (2.17)$$

Coefficient of power is given by

$$C_p = \frac{\text{Rotor power}}{\text{Power in wind}} = \frac{2P}{\rho U_{\infty}^3 \pi R^2} \quad (2.18)$$

By substituting Equation (2.17) into Equation (2.18), the following equation can be obtained

$$C_p = 4a(1 - a)^2 \quad (2.19)$$

In order to find the maximum C_p , Equation (2.19) is differentiated with respect to 'a' and is equated to zero, which gives [9]

$$C_{p,\text{max}} = \frac{16}{27} = 0.5926 \text{ when } a=1/3 \quad (2.20)$$

For an ideal rotor to operate at the point of maximum power production, the wind speed at the rotor should be 2/3 of the freestream velocity of the wind. This is called the Betz limit.

The thrust coefficient on the disk which defines the relationship between the thrust force and dynamic force.

$$C_T = \frac{2T}{\rho U_{\infty}^2 \pi R^2} \quad (2.21)$$

Therefore, from Equations (2.10) and (2.14), the axial thrust is given by

$$T = 2Aa\rho(1-a)U_{\infty}^2 \quad (2.22)$$

By substituting Equation (2.22) into Equation (2.21), thrust coefficient can be written as

$$C_T = 4a(1-a) \quad (2.23)$$

When $a=0.5$, C_T has a maximum value of 1 and U_{wake} is zero. When $a=1/3$ for maximum power production C_T is 8/9. It is evident that this model is not suited for axial induction factor greater than 0.5 because as per the Equation (2.14), U_{wake} will have a negative value. This results in the reverse flow from the wake region into the rotor.

In conclusion Betz limit $C_{P,\text{max}}=16/27$ is the maximum power coefficient. In reality, there are three effects which lead to a decrease in the power coefficient [8].

- Wake rotation behind the turbine
- Finite blade numbers and its associated losses
- Non-zero aerodynamic drag

2.3. Momentum Theory

The axial momentum theory was previously analyzed with the assumption that there is no rotation imparted to the flow in the slipstream (a region behind a moving object in which wake is moving at velocities comparable to the moving object). In general, there will be rotational motion in the slipstream due to the reaction of torque of the rotor as shown in Figure 2.2 and this causes a further loss of energy. The wake rotates in the opposite direction to the rotor. Therefore, the power output is decreased by the rotational kinetic energy of the wake.

Figure 2.2: Wake rotation [8]

Figure 2.3: Annular stream tube control volume [8]

An annular stream tube model is illustrated to explain the rotation of the wake. As shown in Figure 2.3, the stream tube with radius r and thickness dr and cross-sectional area $dA=2\pi r dr$ is considered. The pressure, wake rotation and induction factors are all assumed to be the function of radius. Let ω be angular velocity imparted to the wake by the rotation of the rotor and Ω be the angular velocity of the rotor. It is assumed that ω is small compared to Ω and also far wake pressure equals to the pressure in the freestream region. Therefore, the angular velocity of the air increases from Ω to $(\Omega + \omega)$ across the rotor, while the axial and radial components remain constant. The pressure difference across the blade is

$$p'' = p_{\text{before}} - p_{\text{after}} = \rho \left(\Omega + \frac{\omega}{2} \right) \omega r^2 \quad (2.24)$$

The thrust on each annular ring element is given by

$$dT = p'' dA = 2 \pi \rho \left(\Omega + \frac{\omega}{2} \right) \omega r^3 dr \quad (2.25)$$

By introducing a factor called angular induction factor (a')

$$a' = \frac{\omega}{2\Omega} \quad (2.26)$$

The expression for the thrust is given as

$$dT = 4a'(1+a')\left(\frac{1}{2}\right)\rho \Omega^2 r^2 2\pi r dr \quad (2.27)$$

The thrust on the annular cross section can also be found using the previously discussed linear momentum analysis as given by

$$dT = \left(\frac{1}{2}\right)\rho U_\infty^2 4a(1-a)2\pi r dr \quad (2.28)$$

Equating the two Equations (2.27) and (2.28), the following expression can be obtained:

$$\frac{a(1-a)}{a'(1+a')} = \frac{\Omega^2 r^2}{U_\infty^2} = \lambda_r^2 \quad (2.29)$$

Tip speed ratio is defined as the ratio of blade tip speed to the freestream wind speed.

$$\lambda = \frac{\Omega R}{U_\infty} \quad (2.30)$$

where R is the rotor radius

The local tip speed ratio at a particular intermediate radius r is given by

$$\lambda_r = \frac{\Omega r}{U_\infty} = \frac{\lambda R}{R} \quad (2.31)$$

Assume there is no swirl in the upstream of the blades and in the downstream, there is conservation of angular momentum. Torque is equal to the change in angular momentum of wake. For an annular element, torque dQ is given by

$$dQ = d\dot{m}(\omega r) r = \rho U_{\text{rotor}} 2\pi r dr (\omega r) r \quad (2.32)$$

By substituting $U_{\text{rotor}} = U_\infty(1-a)$ and $a' = \frac{\omega}{2\Omega}$, the following equation is obtained:

$$dQ = 4a'(1-a)\rho U_\infty \Omega r^2 \pi r dr \quad (2.33)$$

Thus, the power generated at each element is given by

$$dP = \Omega dQ \quad (2.34)$$

2.4. Blade Element Theory

In the previous section, the momentum theory is used to determine the thrust and torque of the rotor by applying the conservation of momentum laws. It does not explain the geometry of the rotor like blade aerofoil, chord and twist. It is essential to analyze the blade behaviour due to their rotational motion in the air. In this analysis, blade is considered to be divided into N sections or elements as shown in Figure 2.4 [10, 11]. Further assumptions in Blade Element Theory are:

- The forces on each element are solely determined by the aerofoil characteristics (Lift and Drag Coefficients)
- Each element is independent of another element in the blade. There is no aerodynamic interaction (No flow in the radial direction)

Figure 2.4: Blade element model [6]

Wind velocity at the rotor is $U_\infty (1-a)$ and the circumferential velocity due to rotation of the blade is the sum of blade section velocity Ωr and the induced angular velocity $a'\Omega r$. Therefore, the relative velocity U_{rel} is the vector sum of the axial velocity and the circumferential velocity.

$$U_{rel}^2 = (U_\infty (1-a))^2 + (\Omega r(1+a'))^2 \quad (2.35)$$

From Figure 2.5 and by velocity triangle approach

$$U_{rel} = \frac{U_{\infty}(1-a)}{\sin(\phi)} \quad (2.36)$$

$$\tan(\phi) = \frac{U_{\infty}(1-a)}{\Omega r(1+a')} = \frac{(1-a)}{(1+a')\lambda_r} \quad (2.37)$$

$$\alpha = \phi - \beta \quad (2.38)$$

Figure 2.5: Blade element geometry, velocities and forces [6]

The elemental lift force is given by

$$dF_{lift} = C_L \left(\frac{1}{2} \right) \rho U_{rel}^2 c dr \quad (\text{where } c \text{ is the chord}) \quad (2.39)$$

The elemental drag force is given by

$$dF_{drag} = C_D \left(\frac{1}{2} \right) \rho U_{rel}^2 c dr \quad (2.40)$$

The thrust force and the torque force are found using the force balance.

The elemental torque force is given by

$$dL = dF_{\text{lift}} \sin(\phi) - dF_{\text{drag}} \cos(\phi) \quad (2.41)$$

The elemental thrust force is given by

$$dT = dF_{\text{lift}} \cos(\phi) + dF_{\text{drag}} \sin(\phi) \quad (2.42)$$

If the turbine has 'B' number of blades then the elemental torque force and the elemental thrust force is given by

$$dL = B(dF_{\text{lift}} \sin(\phi) - dF_{\text{drag}} \cos(\phi)) \quad (2.43)$$

$$dT = B(dF_{\text{lift}} \cos(\phi) + dF_{\text{drag}} \sin(\phi)) \quad (2.44)$$

The elemental torque is given by $dQ = r dL$.

By introducing a factor solidity (σ) which is the ratio between the blade area to the area of the disc (blade swept area). It is a non-dimensional factor and it describes the blade geometry.

$$\sigma = \frac{B c}{2 \pi r} \quad (2.45)$$

By substituting Equations (2.36) and (2.45) into the Equations (2.43) and (2.44) and further manipulation gives the general form of elemental thrust and elemental torque equations as

$$dT = \sigma \pi \rho \left(\frac{U_{\infty}^2 (1-a)^2}{\sin^2(\phi)} \right) (C_L \cos(\phi) + C_D \sin(\phi)) r dr \quad (2.46)$$

$$dQ = \sigma \pi \rho \left(\frac{U_{\infty}^2 (1-a)^2}{\sin^2(\phi)} \right) (C_L \sin(\phi) - C_D \cos(\phi)) r^2 dr \quad (2.47)$$

2.5. Blade Element Momentum (BEM) Theory

The Blade Element Momentum theory [12, 13] which is also known as strip theory is used to bridge the gap between the momentum theory and the blade element theory. Assumptions in BEM theory are the combinations of assumptions made in the two theories. One important assumption is that spanwise flow is negligible, therefore, 2D aerofoil data can be used in blade element theory.

By equating Equations (2.28) and (2.46) gives

$$\frac{a}{(1-a)} = \sigma C_L \left(\frac{\cos \phi}{4 \sin^2 \phi} \right) \left[1 + \left(\frac{C_D}{C_L} \right) \tan \phi \right] \quad (2.48)$$

By equating the Equations (2.33) and (2.47) gives

$$\frac{a'}{(1-a)} = \left(\frac{\sigma C_L}{4 \lambda_r \sin \phi} \right) \left[1 - \left(\frac{C_D}{C_L} \right) \cot \phi \right] \quad (2.49)$$

By substituting Equation (2.37) into Equation (2.48) and rearrangement gives

$$\frac{a'}{(1+a')} = \left(\frac{\sigma C_L}{4 \cos \phi} \right) \left[1 - \left(\frac{C_D}{C_L} \right) \cot \phi \right] \quad (2.50)$$

After some algebraic manipulations, the expressions to find the induction factors a and a' are given by

$$a = \left[\frac{4 \sin^2 \phi}{\sigma (C_L \cos \phi + C_D \sin \phi) + 1} \right]^{-1} \quad (2.51)$$

$$a' = \left[\frac{4 \sin \phi \cos \phi}{\sigma (C_L \sin \phi - C_D \cos \phi) + 1} \right]^{-1} \quad (2.52)$$

2.6. Correction Factors for BEM Theory

Traditional BEM theory does not include any losses in the tip and the hub and also it is not suited for the axial induction factor greater than 0.5 as explained in Section 2.2. Following are the important correction factors to be included in the BEM theory:

2.6.1. Prandtl Tip Loss Correction

In BEM analysis, it is assumed that the rotor has an infinite number of blades, but in the actual case, wind turbines have a finite number of blades. There will be changes in wake rotation depending on the blade tip end. This change has an effect in reducing the performance of the rotor [14, 15, 9].

Prandtl who created the concept of boundary layer derived tip loss correction factor F_{tip} , which is given by

$$F_{tip} = \left(\frac{2}{\pi} \right) \cos^{-1} \left(e^{-f_{tip}} \right) \quad (2.53)$$

where,

$$f_{tip} = \left(\frac{B}{2} \right) \left(\frac{R_{tip} - r}{r \sin \phi} \right) \quad (2.54)$$

The correction factor is multiplied into Right Hand Side (RHS) of the Equations (2.28) and (2.33)

$$dT = 4F_{tip} \pi r \rho U_{\infty}^2 a (1-a) dr \quad (2.55)$$

$$dQ = 4F_{tip} \pi r^3 \rho U_{\infty} (1-a) a' dr \quad (2.56)$$

Hence, a and a' will be given as

$$a = \left[\frac{4F_{tip} \sin^2 \phi}{\sigma (C_L \cos \phi + C_D \sin \phi) + 1} \right]^{-1} \quad (2.57)$$

$$a' = \left[\frac{4F_{tip} \sin \phi \cos \phi}{\sigma (C_L \sin \phi - C_D \cos \phi) + 1} \right]^{-1} \quad (2.58)$$

2.6.2. Glauert Correction

The BEM method discussed earlier in this chapter is good for axial induction factor less than 0.4 [15]. But when the wind speed is low there will be relatively large thrust, hence from Equation (2.23), the axial induction factor will also be large.

Based on the experimental results, Glauert [16, 17] derived an empirical correlation that has to be used when $a > 0.4$

Based on Glauert [16, 17] and Spera [18], Equation (2.57) is further modified as

when $a < a_c$, where a_c is the critical value and is equal to 0.2.

$$a = \left[\frac{4F_{tip} \sin^2 \phi}{\sigma (C_L \cos \phi + C_D \sin \phi) + 1} \right]^{-1} \quad (2.59)$$

when $a > a_c$

$$a = 0.5 \left[2 + K_{\text{spera}} (1 - 2a_c) - \sqrt{\left((K_{\text{spera}} (1 - 2a_c) + 2)^2 + 4(K_{\text{spera}} a_c^2 - 1) \right)} \right] \quad (2.60)$$

$$\text{where, } K_{\text{spera}} = \frac{4F_{\text{tip}} \sin^2 \phi}{\sigma (C_L \cos \phi + C_D \sin \phi)} \quad (2.61)$$

2.6.3. Hub Loss Model

Similar to tip loss model, the hub loss model is used for correcting velocity due to the vortex shed near the hub [14]. The hub loss model is identical to Prandtl tip loss model.

$$F_{\text{hub}} = \left(\frac{2}{\pi} \right) \cos^{-1} \left(e^{-f_{\text{hub}}} \right) \quad (2.62)$$

where,

$$f_{\text{hub}} = \left(\frac{B}{2} \right) \left(\frac{r - R_{\text{hub}}}{r \sin \phi} \right) \quad (2.63)$$

2.6.4. Buhl Model

Glauert correction showed instability when both tip and hub losses are included [14, 19, 10]. Buhl [19] derived a new empirical relation between thrust coefficient and axial induction factor to overcome the instability.

$$C_T = \left(\frac{8}{9} \right) + \left(4F - \left(\frac{40}{9} \right) \right) a + \left(\left(\frac{50}{9} \right) - 4F \right) a^2 \quad (2.64)$$

For $C_T \geq 0.96$

$$a = \frac{18F - 20 - 3\sqrt{(C_T(50 - 36F) + 12F(3F - 4))}}{36F - 50} \quad (2.65)$$

For $C_T < 0.96$

$$a = \left[\frac{4F \sin^2 \phi}{\sigma (C_L \cos \phi + C_D \sin \phi) + 1} \right]^{-1} \quad (2.66)$$

where,

$$F = F_{\text{tip}} F_{\text{hub}} \quad (2.67)$$

2.7. BEM algorithm

Figure 2.6 shows the flow chart for analyzing the performance of the given wind turbine using BEM theory. When the blade geometry parameters like aerofoil shape, chord, and twist distribution along the blade length are known, BEM can be used to predict the power generated by the turbine. This requires an iterative process in which the induction factors at each blade element are initially guessed and then determined within an acceptable tolerance of the previous guess during iteration. Flow angle and aerofoil coefficients are determined in each guess during the iteration progress for each blade element until an acceptable tolerance is reached. MATLAB code is written based on this flow chart. Software Qblade is used to validate the generated BEM code.

Figure 2.6: Flowchart for BEM analysis

2.8. Unsteady Aerodynamics and 3D Aspects

Assumptions discussed in Sections 2.2 and 2.4 are ideal. In real time, wind turbines operate in completely complex and unsteady conditions due to atmospheric turbulence, wind shear, yaw misalignment, presence of tower structure etc. Unsteady effects result in dynamic stall,

dynamic inflow and stall delay etc. These unsteady aspects and the models for some aspects are well reported in Hansen and Butterfield [3], Leishman [20] and Hansen [5].

(i) Yaw misalignment refers to the rotating blades experiencing cyclic load change due to the flow being not perpendicular to the turbine. The classical BEM method does not account for non-axial flow. Therefore, it cannot directly be used to analyze the flow in yawed condition of the rotor. But in reality, the model needs to be extended to analyze the yawed flows because many turbines in recent years operate in some degrees of the yawed condition. When the turbine operates in the yawed condition it experiences some cyclic variation in the incidence which may also result in a stall condition. Attempts have been made to resolve these problems and researchers [21-23, 14] have prescribed empirical relations for induction factor in the azimuthal direction. Sant [24] as a part of his thesis work has used free wake vortex model to study the effect of yaw on the NREL Phase VI Turbine sequence S (yaw angle: 30°) and TUDelft Model turbine (yaw angle: 30° and 45°).

(ii) Wind shear refers to the change in the velocity being varied with altitude based on the atmospheric boundary layer. The assumption of wind velocity to be uniform will not be valid and also aerodynamic forces will not be balanced in the presence of wind shear.

(iii) Tower shadow refers to the fluctuation experienced by the downstream turbine blades due to the wake as it comes in line with the tower of the turbine in the front row. According to the classical BEM theory, the inflow velocity is uniform. But the downstream wind turbine might experience reduced local velocity as it passes the tower of the turbine in front. The influence of the tower shadow on the blade has potential to trigger the dynamic stall and also causes unsteadiness even when the flow is attached once in every cycle.

(iv) Dynamic inflow refers to the delay in achieving the steady state when there are sudden fluctuations in the pitch angle, wind or rotor speed. BEM method is time independent and the forces on the blade elements are constant in time. According to the classical BEM theory any changes in the load of the wind turbine, the induced velocity will react instantaneously. Snel and Schepers [22] and Schepers and Snel [23] conducted an in-depth study on the effects of dynamic inflow. Progress on the yaw misalignment effect

and dynamic inflow in comparison with the recent measurements from the NREL Phase VI rotor and MEXICO experiment are provided in detail in the work of Schepers [25].

(v) Dynamic stall refers to the unsteadiness in stalling due to the aerofoil subjected to rapid change like pitching, which results in an immediate change in angle of attack. When the angle of attack suddenly changes from below the angle of static stall to above, flow on the upper surface which is normally separate in the static case will remain attached resulting in increase in lift coefficient. The flow then suddenly separates causing decrease in lift coefficient and increase in drag coefficient. In wind turbines, the angle of attack changes due to yaw misalignment, tower shadow, wind shear, atmospheric turbulence. Leishman and Beddoes [26], Øye [27], Hansen et al. [28], Pereira et al [29] and Liu et al. [30] have studied dynamic stall in detail and have included models for dynamic stall in the BEM analysis. Also, Guntur et al. [31] in recent years have studied the effect of dynamic stall using NREL Phase VI N sequence, dynamic stall models and CFD analysis.

(vi) Stall delay refers to the rise in lift coefficients for the rotating blade when compared to 2D analysis due to the spanwise flow. In BEM analysis, only the axial flow is considered and the spanwise flow along the blade is neglected (radial independence). It is more effective at inboard sections and gradually reduces towards the tip. In general, 2D aerodynamic coefficients obtained either from the experiments or the CFD analyses are used in BEM. This generally results in inaccurate predictions of the performance analysis. The difference between dynamic stall and stall delay is that due to dynamic stall, lift increases instantaneously and drops while in the stall delay the rotation of the blade causes the separation to be delayed causing an increase in lift compared to the values predicted using 2D analysis. The lift does not drop immediately and remains almost fixed. Therefore, BEM needs to be improved and it can account for stall delay and later it should be modelled for other unsteady effects like dynamic stall.

Since in this thesis, NREL Phase VI Turbine (S sequence, zero yaw angle) is analyzed and objective is to provide a pathway to couple between BEM and CFD, only stall delay is focused on as it is an important unsteady aspect which has not been clearly modelled yet.

BEM is a rotor aerodynamics analysis model which needs to be coupled with CFD to understand the effect of wind shear, atmospheric turbulence and tower effect. In order to analyze this effect, the NREL Phase VI Turbine is used and their specifications and CFD analyses are discussed in Chapter 5 for complete rotor analysis in different atmospheric location sites.

2.9. Stall delay or Rotational Augmentation

There is a rise in lift coefficients for rotating blade when compared to 2D analysis due to the spanwise flow, which results in the prevention or delay in the stall occurrence. This phenomenon is called stall delay. Lag in understanding the effect of stall delay may result in the under-prediction of aerodynamic loads and also in design failure. It is more effective at inboard sections and gradually reduces towards the tip.

2.9.1. Previous Observations

In the 1940s, Himmelskamp [32] was the first to notice the difference in the lift coefficients obtained from rotating and non-rotating aeroplane propellers. Banks and Gadd [33] performed a theoretical analysis and explained about how laminar separation is delayed by rotation and concluded that the rotation causes the boundary layer to get stabilized and prevent separation. Dwyer and McCroskey [34] observed and analyzed the stall delay on helicopter rotor blades and later Milborrow [35] and Madsen and Christensen [36] in wind turbines. Ronsten [37] in his experimental analysis illustrated the difference in the lift coefficient measured from the rotating and non-rotating blades. Figure 2.7 shows the lift coefficient measured at 30% of blade span.

In the early 1990s, with improvements in computer technologies, Wood [38], Narramore and Vermeland [39] and Snel et al. [40] performed CFD analysis to understand the effect of stall delay. In 2001, NREL conducted a blind test comparison based on the wind tunnel experiment analysis of the NREL Phase VI Turbine. Researchers were told to find the loads on the NREL Phase VI Turbine without providing them experimental results [41]. On

comparison with the experimental results and predicted values, huge uncertainties were noted.

Figure 2.7: Comparison of lift coefficient for rotating (RB) and non-rotating (NRB) blades at 30% blade span [37]

Therefore, it was concluded that existing models used to adjust the 2D aerofoil characteristics to account for stall delay were incorrect. In 2002, Schreck and Robinson [42] used CFD along with NREL Phase VI experiment results to understand the effect of the radial pressure gradient in the mechanism of stall delay. In 2008, Breton et al. [43] used lifting line prescribed wake vortex theory in order to simulate NREL Phase VI to study the effect of different stall delay models and concluded that existing models result in over-prediction.

2.9.2. Mechanism responsible for stall delay

Stall delay is a boundary layer effect which is predominantly believed to be due to the centrifugal pumping action or dynamic pressure gradient or a combination of both. Both these actions induce spanwise flow, which results in the appearance of the Coriolis force (It is a force experienced by a body or object which is moving in relative to a rotating reference frame. This force is perpendicular to both the rotation axis and the direction of the body's motion) as shown in Figure 2.8 [44]. This Coriolis force acts as a positive

chordwise pressure gradient from the leading edge to the trailing edge and resulting in the stall delay as shown in Figure 2.9 [45]. When the flow is attached at low wind speeds like 7m/s as shown in Figure 2.10, flow remains on the suction side for a short duration as there is no occurrence of stall. Even though the effect of centrifugal action and the dynamic pressure gradient is present, they are minor in action. But when stall occurs at higher wind speeds like 10m/s as shown in Figure 2.11, separated flow stays along the suction surface and experiences centrifugal pumping, hence moves radially [46]. Figures 2.10 and 2.11 are based on the CFD analysis of NREL Phase VI Turbine, which will be further explained in Chapter 5. It is reproduced here for the understanding of stall delay.

Figure 2.8: Stall delay Phenomenon [44]

Figure 2.9: Favourable pressure gradient due to the Coriolis force [45]

Figure 2.10: Limiting streamlines on the suction side for wind speed 7m/s along with the pressure contour on the blade

Figure 2.11: Limiting streamlines on the suction side for wind speed 10m/s along with the pressure contour on the blade

Coriolis force [44] experienced by the separated flow is given by

$$\vec{F}_{\text{Coriolis}} = -2m\vec{\Omega} \times \vec{V} \quad (2.68)$$

$\vec{V}$ is velocity vector acting radially outward direction (m/s)

$\vec{\Omega}$ is angular velocity vector normal to the rotor plane (rad/s)

m is the mass of subjected object (kg)

According to the right-hand rule, Coriolis force acts perpendicular to both these vectors as shown in Figure 2.8, thus accelerating the flow from leading edge to trailing edge, creating a positive pressure gradient and consequently delaying the stall.

(i) *Effect of centrifugal force* [45,47-49]: Separated air flow at the vicinity of trailing edge on the suction side of the rotor experiences centrifugal force due to rotation. Because of centrifugal force, separated flow moves in the spanwise direction. This radial component of velocity acting in a rotating reference frame induces positive chordwise pressure gradient due to Coriolis force. Centrifugal force is considered to be the predominant cause of the stall delay. The magnitude of the centrifugal force is given by

$$F_{\text{centrifugal}} = m\Omega^2 r \quad (2.69)$$

(ii) *Effect of dynamic pressure gradient* [45, 49]: Since the relative wind velocity increases along the radial direction due to the blade rotation, dynamic pressure increases in the radial direction.

Local dynamic pressure [50, 51] at any blade section is given by the equation

$$p_{\text{dynamic}} = 0.5 * \rho * (U_{\infty}^2 + (r * \Omega)^2) \quad (2.70)$$

As dynamic pressure radially increases, there will be a decrease in the static pressure. Separated flow experiences a static pressure difference along the blade span resulting in a spanwise flow from the high static pressure region at inboard to the low static pressure region at the top. As mentioned earlier, when there is a spanwise flow in a rotating reference frame, Coriolis effect occurs resulting in stall delay.

2.9.3. S809 aerofoil characteristics and polar extrapolation

BEM theory requires C_L and C_D as a function of angle of attack (AoA). Experimentally predicted C_L and C_D for S809 aerofoil from various wind tunnel tests conducted at Ohio State University (OSU), Colorado State University (CSU) and Delft University of Technology (DUT) for different ranges of Reynolds numbers are available in the report [8]. Figure 2.12 (a and b) shows C_L and C_D as a function of AoA and Reynolds number from the experimental results mentioned in the reference [8]. Values of C_L and C_D remain close for all measurements of the lower angles of attack, but at higher angles of attack, there are differences even for the same Reynolds number of experiments in different universities. Sørensen et al. [52] indicated that Reynolds number is in the range 0.7×10^6 to 1.4×10^6 near the root and around 1.0×10^6 to 1.1×10^6 near the tip for NREL Phase VI Turbine S sequence.

(a)

(b)

Figure 2.12: Aerofoil characteristics of S809 aerofoil from various experiments [8]. (a) lift coefficients of S809 aerofoil (b) drag coefficients of S809 aerofoil

Many researchers including Lanzafame and Messina [10] have also chosen the values of C_L and C_D from DUT wind tunnel analysis for Reynolds number of 1×10^6 . Since the values of C_L and C_D are only available up to 20.6° AoA from the experiment, there are various methods employed for extrapolation as follows:

2.9.3.1 Viterna and Corrigan model extrapolation [53]

Viterna and Corrigan [53] developed an empirical correlation for extrapolation using the provided aspect ratio (AR) and stall angle of attack (AoA) α_s with its aerofoil characteristics as follows:

$$C_L = \frac{C_{D,\max}}{2} \sin(2\alpha) + K_L \frac{\cos^2(\alpha)}{\sin(\alpha)} \quad (2.71)$$

$$C_D = C_{D,\max} \sin^2(\alpha) + K_D \cos(\alpha) \quad (2.72)$$

$$K_L = (C_{L,s} - C_{D,\max} \sin(\alpha_s) \cos(\alpha_s)) \frac{\sin(\alpha_s)}{\cos^2(\alpha_s)} \quad (2.73)$$

$$K_D = \frac{C_{D,s} - C_{D,\max} \sin^2(\alpha_s)}{\cos(\alpha_s)} \quad (2.74)$$

$$\text{if } AR \leq 50 : C_{D,\max} = 1.11 + 0.018(AR) \quad (2.75)$$

$$\text{if } AR > 50 : C_{D,\max} = 2.01 \quad (2.76)$$

The important parameter necessary for this model is the aspect ratio. There are different values of aspect ratio (AR) for NREL Phase VI Turbine available in the literature based on different definitions.

Lindenburg [45] has specified AR to be 7.2 based on the expression

$$AR = R^2 / \text{planform area} \quad (2.77)$$

Tangler and Kocurek [54] used AR to be 11 based on the expression

$$AR = R / (\text{chord at } 0.8R) \quad (2.78)$$

But Tangler and Kocurek [54] also suggested using AR as 14 for enhanced results.

Figures 2.13 and 2.14 show the C_L and C_D values of S809 extrapolated using Viterna and Corrigan model [53] from DUT experiment results for Reynolds number 1×10^6 respectively.

Figure 2.13: C_L extrapolation from 0° to 90° for S809 aerofoil using Viterna-Corrigan model [53]

Figure 2.14: C_D extrapolation from 0° to 90° for S809 aerofoil using Viterna-Corrigan model [53]

Figure 2.15 shows the comparison of power computed with Viterna and Corrigan model [53] extrapolation for AR 7.2, 11 and 14 using BEM MATLAB Code and Qblade. At lower wind speeds up to 10 m/s, sectional AoA will be less than 20° and the power predicted has a good closeness to experimental values. At higher wind speeds, where the AoA will be higher, C_L and C_D extrapolated from Viterna and Corrigan model [53] are applied. It is evident from Figure 2.15 that power is over-predicted at higher wind speeds.

Figure 2.15: Power curves using Viterna-Corrigan model [53] from BEM MATLAB code (left) and Qblade (Right)

2.9.3.2 Lanzafame and Messina [10] extrapolation proposal for BEM analysis of NREL Phase VI Turbine

As mentioned earlier, Lanzafame and Messina [10] conducted BEM analysis with DUT wind tunnel tests up to 20.6° AoA and for the AoA above 20° region is classified as a flat plate fully stalled regime. They suggested the following expressions:

$$C_L = 2C_{L,max} \sin(\alpha) \cos(\alpha) \quad (2.79)$$

$$C_D = C_{D,max} \sin^2(\alpha) \quad (2.80)$$

where $C_{L,max} = C_L$ at 45° and $C_{D,max} = C_D$ at 90°

There is no clear derivation or reference provided for these expressions. It is assumed in the present analysis that Lanzafame and Messina [10] had derived the Expression (2.79) and (2.80) from Viterna and Corrigan model [53] on the assumption of the flat plate theory for AoA >20° and stall angle of attack for flat plate to be 45° as explained as follows.

For a flat plate [54] lift and drag coefficients are provided as follows,

$$C_L = 2\sin(\alpha) \cos(\alpha) \quad (2.81)$$

$$C_D = 2\sin^2(\alpha) \quad (2.82)$$

Figure 2.16 shows C_L and C_D versus AoA for a flat plate plotted based on Equations (2.81) and (2.82) respectively and at it can be noticed at 45° AoA, $C_L = C_D = 1$.

Figure 2.16: C_L , C_D vs AoA for a flat plate [54]

For a flat plate, following expressions can be obtained

$$C_{L,max}=C_L \text{ at } 45^\circ \quad (2.83)$$

$$C_{D,max}=C_D \text{ at } 90^\circ \quad (2.84)$$

$$C_{D,max}=2 \times C_D \text{ at } 45^\circ \quad (2.85)$$

$$C_{D,max}=2 \times C_L \text{ at } 45^\circ \quad (2.86)$$

$$C_{D,max}=2 \times C_{L,max} \quad (2.87)$$

Since C_L starts to decrease after 45° AoA, Lanzafame and Messina model [10] might have assumed the stall angle for flat plate is 45° and derived the following expressions

$$\begin{aligned} (C_{D,max} \sin(\alpha_s) \cos(\alpha_s)) &= 2C_{L,max} \sin(\alpha_s) \cos(\alpha_s) \\ &= 2C_{L,s} \frac{1}{\sqrt{2}} \frac{1}{\sqrt{2}} \\ &= C_{L,s} \end{aligned} \quad (2.88)$$

$$\begin{aligned} C_{D,max} \sin^2(\alpha_s) &= 2C_{D,s} \left(\frac{1}{\sqrt{2}} \right)^2 \\ &= C_{D,s} \end{aligned} \quad (2.89)$$

Combination of C_L expression based on Viterna and Corrigan Model [53] and flat plate model provides

$$C_L = \frac{C_{D,max}}{2} \sin(2\alpha) + K_L \frac{\cos^2(\alpha)}{\sin(\alpha)} \quad (2.90)$$

$$C_L = \frac{C_{D,max}}{2} 2\sin(\alpha)\cos(\alpha) + K_L \frac{\cos^2(\alpha)}{\sin(\alpha)} \quad (2.91)$$

$$C_L = 2C_{L,max} \sin(\alpha)\cos(\alpha) + K_L \frac{\cos^2(\alpha)}{\sin(\alpha)} \quad (2.92)$$

By substituting Equation (2.88) for flat plate in Equation (2.73) results in $K_L=0$

Hence, Equation (2.92) becomes

$$C_L = 2C_{L,max} \sin(\alpha)\cos(\alpha) \quad (2.93)$$

Combination of C_D expression based on Viterna and Corrigan Model [53] and Flat plate model provides

$$C_D = C_{D,\max} \sin^2(\alpha) + K_D \cos(\alpha) \quad (2.94)$$

By substituting Equation (2.89) for flat plate in Equation (2.74) results in $K_D=0$

Hence, Equation (2.94) becomes

$$C_D = C_{D,\max} \sin^2(\alpha) \quad (2.95)$$

This derivation is purely based on my own intellectual since there is no proper derivation or reference provided by Lanzafame and Messina [10].

Nevertheless, these expressions need C_L at 45° AoA and C_D at 90° AoA. Table 2.1 shows different C_L at 45° and C_D at 90° predicted for different Reynolds numbers from the reference [8]. Figures 2.17 and 2.18 show C_L and C_D for S809 aerofoil from 0° to 90° AoA respectively using Equations (2.79) and (2.80) and values in Table 2.1.

Reynolds number	$C_{L,\max}$ (C_L at 45°)	$C_{D,\max}$ (C_D at 90°)
300,000	1.230	2.240
500,000	1.200	2.090
650,000	1.120	2.270

Table 2.1: Different C_L at 45° and C_D at 90° predicted for different Reynolds numbers from the reference [8]

Figure 2.17: C_L extrapolation from 0° to 90° for S809 aerofoil using Lanzafame and Messina method [10]

Figure 2.18: C_D extrapolation from 0° to 90° for S809 aerofoil using Lanzafame and Messina method [10]

Figure 2.19 reveals the comparison of power computed using BEM MATLAB code and Qblade via Lanzafame and Messina proposal [10] with values of C_L and C_D taken from three different Reynolds numbers. It can be noticed that Lanzafame and Messina [10] showed good concordance with experimental results on choosing the values of C_L and C_D for $Re = 5 \times 10^5$ at lower wind speed (7m/s) and higher wind speeds (20 and 25 m/s). A similar closure is obtained on choosing the values for $Re = 3 \times 10^5$. But when the values of $Re = 6.5 \times 10^5$ are used there is a deviation in power prediction at moderate and higher wind speeds. It is also noticed that though Lanzafame and Messina [10] showed the closure

results for power, but on comparing the sectional aerofoil coefficients with the 3D values obtained from inverse BEM (explained later in this chapter) indicates clear deviation. This method has two drawbacks; one is the need of appropriate aerofoil characteristics at 45° and 90° and the second is sectional aerofoil characteristics are not calculated correctly, therefore, it does not account for stall delay. The second point will be clearly explained in Section 6.3 of Chapter 6 on comparing C_L distribution along the blade with C_L along the blade span computed from CFD results

Figure 2.19: Power curves using Lanzaforme and Messina [10] proposal from BEM
MATLAB code (left) and Qblade (Right)

2.9.3.3. Tangler and Kocurek [54] extrapolation proposal for BEM analysis of NREL Phase VI Turbine

Tangler and Kocurek [54] have used Lifting Surface Wake Theory (LSWT) in determining AoA and then computed $C_{L,3D}$ and $C_{D,3D}$ from experimental computed C_{NOR} and C_{TAN} of NREL Phase VI Turbine for wind speed range from 5 to 20 m/s at five different radial locations (0.3R, 0.47R, 0.63R, 0.80R, 0.95R). From the derived data, Tangler and Kocurek [54] expressed that after 20° AoA, the relationship of AoA with C_L and C_D is similar to the flat plate expression. Therefore, C_L and C_D at different radial sections are averaged up to

20° and the expression for flat plate has been used for computation of C_L and C_D from 20° to 90° which is given as follows:

$$C_L = 2\sin(\alpha)\cos(\alpha) \quad (2.96)$$

$$C_D = 2\sin^2(\alpha) \quad (2.97)$$

In Equations (2.73) and (2.74) of Section 2.9.3.1: Viterna and Corrigan model [53] extrapolation, stall AoA and their corresponding aerofoil characteristics were used. Tangler and Kocurek [54] also suggested modification by replacing stall AoA with AoA at which similarity with for flat plate expressions commences (20° AoA for this case as mentioned previously) and its corresponding C_L and C_D instead of $C_{L,s}$ and $C_{D,s}$ in Equations (2.73) and (2.74) respectively. Figure 2.20 shows C_L and C_D versus AoA extracted from Tangler and Kocurek [54] to be used in BEM program and Qblade.

Figure 2.21 shows the power comparison using BEM code and Qblade with C_L and C_D as provided by Tangler and Kocurek [54]. Tangler and Kocurek [54] used WT_Perf BEM software from NREL. NREL no longer supports WT_Perf and removed it from their website. By using C_L and C_D values suggested by Tangler and Kocurek [54] in BEM MATLAB code and Qblade, similar results were obtained as seen in Figure 2.21. However, these results did not correspond to the results shown by Tangler and Kocurek [54] in their work. Since WT_Perf was however no longer available, we were unable to perform the validation using WT_Perf.

Figure 2.20: C_L and C_D extrapolation from 0° to 90° for S809 aerofoil using Tangler and Kocurek proposal [54].

Figure 2.21: Power curves using Tangler and Kocurek [54] proposal from BEM MATLAB code (left) and Qblade (Right)

A strange thing was noticed in the work of Tangler and Kocurek [54] and the Figures 6 and 9 in their work are reproduced here in Figure 2.22 (a) and (b) respectively.

(a)

(b)

Figure 2.22: Figures extracted from Tangler and Kocurek [54]. (a) Figure 6 (C_L and C_D versus wind speed) (b) Figure 9 (C_L and C_D versus Angle of Attack)

It can be noticed in Figure 2.22 (a and b) that 30% blade span plot (marked in yellow in legends) has the same lift and drag coefficients for both wind speed and angle of attack.

This indicates that wind speed and angle of attack have a linear relationship at 30% blade span. For example, at 20 m/s wind speed, the angle of attack at 30% blade span is 20°. This is not feasible, as with the increase in wind speed, the angle of attack will be higher as can be noticed in the plot of streamline in Figure 2.23 which is based on CFD analysis of NREL Phase VI Turbine at 20 m/s. The set up for CFD analysis will be explained in Chapter 5. Full description on the discussion of the results of CFD analysis will be explained in Section 6.1 of Chapter 6. It can also be noticed that the results predicted by Tangler and Kocurek [54] deviate from other researchers' works at 30% blade span in Figure 6.16 of Chapter 6. This error might be due to some editing during their calculations.

Figure 2.23: Streamline along the section 30% of blade span for wind speed 20 m/s.

2.9.4. 3D stall delay correction models

Different empirical models have been developed in the last few decades to correct the 2D aerofoil C_L and C_D values for stall delay. These models incorporate a correction term being added to the 2D characteristics. In general, this additional term is a function of the chord/radius and a few other parameters. The formulation is given as follows:

$$C_{L,3D} = C_{L,2D} + f_{CL}(2\pi(\alpha - \alpha_o) - C_{L,2D}) \quad (2.98)$$

$$C_{D,3D} = C_{D,2D} + f_{CD}(C_{D,2D} - C_D(\alpha = 0)) \quad (2.99)$$

In 1994, Snel et al. [40] proposed a basic model for the correction of lift coefficient to account for stall delay. This model does not include the correction for drag. Snel et al. [40] performed order of magnitude physics on the boundary layer equations to derive a simple model. In 1998, Du and Selig [55] performed an analysis on three-dimensional boundary layer equations in the form of integrals and proposed an extension of the Snel et al. [40] model. Correction for drag coefficient is also taken into consideration. In 2000, Chaviaropolous and Hansen [56] used simplified quasi 3D model to find the effect of rotation on the sectional aerofoil characteristics. This model uses blade twist angle as a function in addition to (c/r) .

Lindenburg [47] in 2004 analyzed the centrifugal and Coriolis force acting on the separated flow instead of boundary layer equations. This model also does not include the correction for the drag coefficient. Table 2.2 shows different stall delay models and their corresponding f_{CL} and f_{CD} expressions.

Figure 2.24 presents the 3D correction models applied to BEM with aerofoil characteristics extrapolation obtained using Viterna and Corrigan model [53], Lanzafame and Messina model [10] and Tangler and Kocurek [54] model. It can be noticed that these models have good predictions at low wind speeds. But as wind speed increases, the power values are over-predicted. The only comparison of power is provided in this section to understand the drawbacks of these stall delay models.

Prediction of aerofoil coefficients along the blade span and its comparison with the results from the NREL Phase VI Turbine and their behaviour will be discussed in Chapter 6. Appendix A includes the BEM program with tip loss, hub loss and Buhl correction factors and incorporates all aspects discussed above. A suitable method can be selected as required.

Models	f_{CL}	f_{CD}
Snel et al. [40]	$f_{CL} = 3 \left(\frac{c}{r} \right)^2$	$f_{CD} = 0$
Du and Selig [55]	$f_{CL} = \frac{1}{2\pi} \left[\left(\frac{1.6 \left(\frac{c}{r} \right)}{0.1267} \right) \left(\frac{1 - \left(\frac{c}{r} \right)^{\frac{R}{\Lambda r}}}{1 + \left(\frac{c}{r} \right)^{\frac{R}{\Lambda r}}} \right) - 1 \right]$ $\Lambda = \Omega R / \sqrt{U_\infty^2 + (\Omega R)^2}$	$f_{CD} = \frac{-1}{2\pi} \left[\left(\frac{1.6 \left(\frac{c}{r} \right)}{0.1267} \right) \left(\frac{1 - \left(\frac{c}{r} \right)^{\frac{R}{2\Lambda r}}}{1 + \left(\frac{c}{r} \right)^{\frac{R}{2\Lambda r}}} \right) - 1 \right]$ $\Lambda = \Omega R / \sqrt{U_\infty^2 + (\Omega R)^2}$
Chaviaropolous and Hansen [56]	$f_{CL} = 2.2 \left(\frac{c}{r} \right) \cos^4(\beta)$	$f_{CD} = 2.2 \left(\frac{c}{r} \right) \cos^4(\beta)$
Lindenburg [47]	$f_{CL} = 3.1 \left(\frac{\Omega r}{U_{rel}} \right)^2 \left(\frac{c}{r} \right)^2$	$f_{CD} = 0$

Table 2.2: Different stall delay models

Figure 2.24: Power curves predicted with different stall delay models

2.10 Inverse BEM

The failure of the proper power prediction by BEM analysis with existing stall delay models and recent development in computing memory emphasizes the need of using the CFD analysis of the wind turbines. It is difficult to determine the effective AoA from experiments or CFD directly. There are several techniques employed to determine the sectional AoA for NREL Phase VI Turbine. Sørensen et al. [52] matched the 2D and 3D stagnation point location to compute sectional lift and drag. Johansen and Sørensen [57] used the calculated average sectional axial induction factor and applied it along with rotational speed and twist to determine the effective AoA. Tangler and Kocurek [54] suggested lifting surface prescribed wake theory (LSWT) to compute the AoA. Sant et al. [58] and Schepers and van Rooij [59] used free wake vortex model developed by the DUT to estimate AoA. Lindenburg [47] and Guntur and Sørensen [60] had used inverse BEM method to compute sectional lift and drag. The methods mentioned above either need pressure measurements at different sections or calculated normal and tangential forces from the pressure measurements. In these previous works, $C_{L,3D}$ and $C_{D,3D}$ had been computed only at 30%, 47%, 63% and 95% of the blade span. Since the objective of this project is to improve the BEM analysis to account for stall delay, the author opted to use the inverse BEM method explained in the work of Lindenburg [47] and Guntur and Sørensen [60], with improvements. In classical BEM analysis, at each element C_L and C_D are provided as an input based on the computed AoA. The provided C_L and C_D are used to calculate the forces acting on that particular element. But in the inverse BEM analysis, forces computed from the CFD analysis are provided as an input to compute the C_L and C_D at that element in the reverse direction of the procedure of the BEM analysis. Figure 2.25 shows the flowchart for the comparison classical BEM analysis of an element and the inverse BEM method by Guntur and Sørensen [60] at each element.

In the work of Guntur and Sørensen [60], only tip loss correction factor was included in the computation of induction factors a and a' . In this present work, in addition to the tip loss

correction factor, hub loss correction factor and correction suggested by Buhl [19] for axial induction factor greater than 0.4 have also been implemented as presented in Figure 2.26. In Figure 2.26, 'F' indicates the product of tip loss (F_{tip}) and hub loss (F_{hub}) correction factors.

Figure 2.25: Comparison of classical BEM and Inverse BEM in an element. (a) classical BEM in an element (b) inverse BEM based on Guntur and Sørensen [60]

The works of Sørensen et al. [52], Johansen and Sørensen [57], Tangler and Kocurek [54], Sant et al. [58] and Schepers and van Rooij [59] have considered only five radial locations where the experimental analysis was performed to compute $C_{L,3D}$ and $C_{D,3D}$. In addition, it is concluded that if the computed pressure measurements at five radial locations from CFD were close to the experimental results along the blade, it is obvious that the pressure

measurements at other locations will also be close to experiments on measurement. In this work, to have an in-depth understanding of stall delay, in addition to 5 locations where the experimental pressure measurement was done, additional 13 locations are provided for developing the geometry for NREL Phase VI Turbine were considered. Therefore, the $C_{L,3D}$ and $C_{D,3D}$ have been computed at 18 locations (0.25R, 0.267R, 0.30R, 0.3275R, 0.39R, 0.45R, 0.47R, 0.51R, 0.57R, 0.63R, 0.633R, 0.691R, 0.75R, 0.80R, 0.81R, 0.87R, 0.93R and 0.95R). Appendix B includes the MATLAB code used for inverse BEM.

Figure 2.27 shows the aerodynamic force coefficients acting on the blade sections. Pressure coefficient C_p at each radial location is computed using the following expression

$$C_p = \frac{p - p_\infty}{\left(\frac{1}{2}\right)\rho_\infty (U_\infty^2 + (\Omega r)^2)} \quad (2.100)$$

From C_p , C_{NOR} and C_{TAN} are calculated by integrating C_p along the suction and the pressure sides of the sectional aerofoil as given as follows

$$C_{NOR} = -\int_0^1 C_{p,u} d\left(\frac{x}{c}\right) + \int_0^1 C_{p,l} d\left(\frac{x}{c}\right) \quad (2.101)$$

$$C_{TAN} = -\int_0^1 C_{p,l} d\left(\frac{y}{c}\right) + \int_0^1 C_{p,u} d\left(\frac{y}{c}\right) \quad (2.102)$$

As shown in Figure 2.27, C_{NOR} is normal force coefficient (normal force acts perpendicular to the chord) while C_{TAN} is tangential force coefficient (tangential force acts along the chord).

From C_{NOR} and C_{TAN} , C_{THRUST} and C_{TORQUE} are computed using the following expressions

$$C_{THRUST} = C_{NOR} \cos(\beta) - C_{TAN} \sin(\beta) \quad (2.103)$$

$$C_{TORQUE} = C_{NOR} \sin(\beta) + C_{TAN} \cos(\beta) \quad (2.104)$$

As shown in Figure 2.27, C_{THRUST} is thrust force coefficient (thrust acts perpendicular to the plane of rotation) while C_{TORQUE} is torque force coefficient (torque force acts along the

plane of rotation). Results and comparisons of computed sectional C_L and C_D for NREL Phase VI Turbine for different wind speeds are discussed in Chapter 6.

Figure 2.26: Flowchart for improved inverse BEM

Figure 2.27: Aerodynamic force coefficients [61]

2.11 Implementation of computed $C_{L,3D}$ and $C_{D,3D}$ in BEM analysis for wind farm

analysis

BEM method plays a vital role in the computation of wind farm analysis. However, the simulation of even a single turbine needs high computational power, memory and time. Thus, researchers suggest different coupled analytical and simulation methods like Actuator Disk model, Actuator Line model and Actuator Surface models for wind farm analysis (briefly discussed in Chapter 7). In these coupled models, forces or pressure drop computed from BEM of rotor are used as input in simulations to reduce the time and memory required. Measurements on improving BEM analysis for stall delay can provide better force prediction which in turn aid in improving the wind farm analysis. In traditional BEM analysis, C_L and C_D are input as a function of AoA computed from 2D wind tunnel or CFD analysis. Since the effect of stall delay on the aerofoil characteristics varies with the rotor radius, C_L and C_D should also be a function of radial locations. It indicates that for the same AoA, two different aerofoil characteristics have to be input at inboard sections and outboard sections of the blade.

Figure 2.28 shows the illustration of the effect of stall delay, at the radial locations 0.25R, 0.267R, 0.63R and 0.633R, AoA was around 25° and the corresponding C_L values are computed as 2.0228, 2.2054, 1.0125 and 1.0239 respectively. From the 2D experiment data by OSU and Reynolds number 1×10^6 for AoA 25° , C_L value is around 0.84. This clearly indicates that there is an increase in the C_L value at inboard sections and it gradually reduces towards the tip. Therefore, the C_L and C_D values in wind turbine analysis in addition to being function of AoA and Reynolds number should also be a function of radial locations because of stall delay.

Figure 2.28: Angle of attack (left) and C_L (right) distributions along the blade span for wind speed 15m/s

Stall delay models discussed in Section 2.9.4 include an additional term to account for this function of radial locations. But these models resulted in over-prediction of power curves at high wind speeds as shown in Figure 2.24. The distribution of C_L and C_D along the radial sections will be discussed in Chapter 6.

In the new proposed BEM analysis, blade is virtually divided into 15 sections with a step of $0.05R$ increment from $0.25R$ to $1.0 R$ as shown in Figure 2.29. Computed C_L and C_D based on the pressure measurement at each span location which falls in a particular section is used for that section. If there are two span locations in a section, the values of aerofoil characteristics are averaged. If no span location falls in a section, aerofoil characteristics of the nearest span location of the adjacent section are used. Figure 2.30 shows the comparison of classical BEM and proposed BEM for the full rotor analysis. Figure 2.30 (a) shows the flow chart of the classical BEM analysis and Figure 2.30 (b) shows the new proposed method for the full rotor. By the proposed model, C_L and C_D are input in BEM analysis as a function of radial locations in addition to be a function of AoA. Computed

results and comparisons are discussed in Chapter 6. Appendix C includes the MATLAB code for the proposed BEM method.

Figure 2.29: Method for distribution of 3D aerofoil characteristics computed from pressure measurements.

(a)

(b)

Figure 2.30: Comparison of classical BEM and proposed BEM. (a) classical BEM (b) proposed BEM

CHAPTER 3: WAKE AERODYNAMICS AND MODELLING

3.1. Atmospheric Boundary Layer Modelling

3.1.1. Introduction

In order to analyze the wake aerodynamics, it is necessary to study the nature of the wind in different terrains and their corresponding atmospheric boundary layer as it directly influences the velocity deficit and turbulence in the wake of the wind turbine. These characteristics of the wind turbine wake are explained in detail in the forthcoming sections of this chapter. In this section, introduction to atmospheric boundary layer and their velocity profiles and turbulent properties are discussed which are used in CFD analysis of NREL Phase VI Turbine in different terrains especially at inlet boundary conditions in order to model fully developed ABL. Details of this CFD analysis are discussed in Chapter 5.

Wind is the movement of air due to the pressure gradient. Solar radiations are the primary source of the wind flow. It is due to the temperature difference caused by the non-uniform heating of the earth's surface by the sun. For example, in the daytime land gets heated more than sea, therefore the air in the land gets warmer and moves upward to higher altitude creating a low pressure region at lower altitudes, cold air at high pressure above the sea zone at low altitudes moves to the low pressure region in the land resulting in sea breeze. While at night, the reverse happens resulting in the movement of air from land to sea which is called land breeze [1].

3.1.2. Geostrophic wind

Similarly, wind circles around the earth due to the uneven heating from the equator to the two poles. Air at the equator is heated to a higher temperature and it moves vertically upward creating a low pressure region near to the surface. The air at higher latitudes near to the poles which are less heated by the sun tends to move towards the equator region. As the air moves, it is subjected to Coriolis force due to the earth rotation. As discussed in Chapter 2, Coriolis force acts perpendicular to the rotation axis (earth axis) and direction of body (air) motion. In the northern hemisphere wind is deflected to the right and in the

southern hemisphere wind is deflected to the left due to the Coriolis force. The deflection increases with an increase in wind speed and terminates when Coriolis force is balanced with the pressure gradient force. This is known as geostrophic balance and the wind in this state is called as geostrophic wind. Geostrophic wind flows in parallel to isobars. In real time, winds would not be exactly geostrophic due to other forces like friction on the earth's surface. But wind tends to have good approximation and closure to the theoretical geostrophic wind as altitude increases and the effect of friction reduces to almost negligible. This region can be considered as the free atmosphere and the height at which it starts depends on the height of the atmospheric boundary layer which is described in Section 3.1.3 [2-4]. Figure 3.1 shows the atmosphere above the earth's surface [5].

Figure 3.1: Atmospheric boundary layer and free atmosphere above earth's surface [5]

3.1.3. Atmospheric boundary layer (ABL)

Atmospheric or planetary boundary layer is the region above the earth's surface (at which the velocity is zero) to the height at which velocity reaches 0.99 of the free atmosphere velocity. On approaching towards the earth's surface from the free atmosphere inside the ABL as going down, the effect of the Coriolis force reduces and the effect of friction force dominates. It is well known that friction depends on the roughness of the earth's surface. Hence the roughness of the terrain is the predominant factor in influencing the height of the ABL, its turbulence and vertical variation (increase) of the velocity still to the free atmosphere. Figure 3.2 shows the effect of roughness on the velocity profile of the ABL [6].

As shown in Figure 3.2, ABL is thicker in rougher surfaces like city, forest and thickness gradually decrease with the roughness and becomes thinner in smooth surface such as the open sea. Roughness of the different terrains are characterized by a value called roughness length (z_0). ABL can be divided again into two regions. They are surface layer and Ekman layer. The surface layer is lower and closer to the earth's surface and its height varies from 20m to 100m depending on the atmospheric stability (described in the next section) and the effect of friction is predominant while Coriolis effect is almost negligible. Above the surface layer is Ekman layer, in which effect of friction gradually reduces and effect of Coriolis force increases and it extends still to the free atmosphere and its height varies from a few hundred meters to around 1km [7]. Figure 3.3 shows the effect of friction with increasing altitude [8]. Wind energy for the wind turbine is extracted mainly from the wind in the surface layer. Table 3.1 shows values of the different ranges of roughness length for different terrains [7].

Figure 3.2: Variation of atmospheric boundary layer height for different terrains [6]

Figure 3.3: Effect of friction with ABL height [8]

Type of terrains	Roughness Length z_0 (m)
City side, forests	0.7
Wooded countryside, suburbs,	0.3
Village side	0.1
Farmlands, areas with few buildings and trees	0.03
Grassy plains	0.01
Desert and rough sea	0.001

Table 3.1: Important types of terrains and their corresponding roughness length [7]

3.1.4. Thermal stratification

Development of ABL (surface layer) in a terrain is also affected by the temperature gradient due to the heating and cooling of the earth's surface and also on the presence of clouds.

Based on thermal stratification, ABL is classified as

(i) *Unstable boundary layer*

This occurs at day time. Due to the heating of the earth's surface by the sun, the surface is warmer than the atmosphere. Due to the temperature difference, buoyancy force occurs due to which warmer air near the surface moves upward towards higher altitude and cold air moves downward to lower altitude. As a result, there will be large vertical motion of the air in the atmosphere and boundary layer thickness increases to 1 to 2 km and heat flux is upwards [9, 10].

(ii) *Stable boundary layer*

This occurs during the night time and the surface is cooler than the atmosphere. The air density is higher near the ground and the vertical motion is damped and wind shear is higher. The thickness of this layer will be in few hundred meters and the heat flux is downwards [9, 10].

(iii) *Neutral boundary layer*

This occurs during some portion of the morning and the evening (between stable and unstable condition). Heat flux is almost zero and thermal effect is negligible or less significant. This layer is easier to model in a wind tunnel and it is close to the condition with strong clouds and the effect of surface heating and cooling are less significant [9,10]. Figure 3.4 shows the velocity profile of the wind speed for unstable, stable and neutral boundary layer [10].

In this work, the effect of temperature and upward motion of the wind due to buoyancy is not included as it is left for future work. Emphasis is made on the neutral ABL and equations described in the following sections are corresponding to the neutral ABL and User Defined Function (UDF) to be used in CFD analysis was written (attached in Appendix D) based on the velocity profile equations, turbulence kinetic energy, turbulent dissipation rate and roughness length as described Section 3.1.5.

Figure 3.4: Vertical velocity profile for unstable, stable and neutral boundary layer [10]

3.1.5. Modelling of neutral homogenous ABL

In neutral ABL, thermal effect is neglected, the occurrence of turbulence is only due to the wind shear. Richards and Hoxey [11] proposed the following equations for the fully developed horizontally homogenous ABL for the $k-\varepsilon$ turbulence model. $k-\varepsilon$ turbulence model is described briefly in Chapter 4.

Governing equation of the mean wind speed in neutral ABL [11-13] is given by

$$\bar{u}(z) = \frac{u_*}{K} \ln \left(\frac{z + z_o}{z_o} \right) \quad (3.1)$$

where,

z is the height above the ground (m)

K is von Karman constant (=0.4)

u_* is friction velocity (m/s)

z_o is the roughness length of the terrain (m)

Turbulence kinetic energy [11-13] is given by

$$k = \frac{u_*^2}{\sqrt{C_\mu}} \quad (3.2)$$

where,

C_μ is a constant of the k- ε turbulence model (In ANSYS FLUENT, its value is 0.09)

Turbulent dissipation rate [11-13] is given by

$$\varepsilon = \frac{u_*^3}{K(z + z_o)} \quad (3.3)$$

In ANSYS FLUENT, roughness length is applied to the ground using the term roughness height and their relationship is provided by

$$k_s = \frac{9.793 * z_o}{C_s} \quad (3.4)$$

where,

k_s is roughness height (m)

C_s is roughness constant

These parameters are used to create fully developed inlet homogeneous boundary conditions for wind turbine wake analysis on different terrains. Details of values of these parameters used in this work, mesh and boundary conditions are discussed in Chapter 5 in CFD analysis of NREL Phase VI Turbine in different terrains. Based on experience, it is noted that inappropriate modelling of ABL and roughness might lead to decay in turbulent characteristics and it affects the characteristics of velocity recovery and turbulence in the wake region. It is necessary to emphasize the importance of the ABL modelling in wake analysis.

3.2. Wake Aerodynamics

3.2.1. Introduction

There are numerous factors that influence the performance of the wind turbines in the large wind farm that covers a large area of land (onshore) or water (offshore). The most important

study in the arrangement of wind turbines in the wind farm is the effect of wake interaction. Thus, wake aerodynamics is one of the predominant topics of research in wind turbines [14-25]. Figure 3.5 shows the wake's photograph caught by a pilot from the helicopter's window focusing on the offshore wind farm in Horns Rev 1 on 12th February 2008 [26].

The two main characteristics [23] of the wind turbine wake are the:

1. Significant **velocity deficit** which results in the lower power extraction in the downstream turbines of the wind farm;
2. Enhanced **turbulence levels**. This can cause increased fatigue loads and on the downstream turbine and thus reduced life span of it.

Figure 3.5: Wind farm wake -Danish offshore wind farm “Horns Rev 1” [26]

There are various parameters such as inflow aerodynamics (discussed in the previous section), design of blades, rotational speed that can produce influence on the structure of the wakes. The most important phenomenon in understanding the structure of the wake is the rotation of the wake in the downstream of the turbine. As discussed in the previous chapter about blade element momentum analysis, due to the rotation of rotor there will be a circumferential component of velocity induced in the downstream of the wind turbine. This

circumferential component initiates the rotation of the wake in the counter direction of the rotor rotation. The flow is in the opposite direction due to the torque exerted by the turbine. Another predominant fact in the investigation of the structure of the wake is the formation of tip vortices. Wake flow will be at relatively low speed compared to the freestream ambient and highly turbulent flow. The tip vortices formed will create a shear layer that separates low speed wake from the freestream ambient flow. As the air flows around the blade profile, it gets accelerated on the suction side. Aerodynamic lift is generated between the two sides (suction side and pressure side) due to the difference in pressure which in turn causes the blades to rotate.

In order to even out the pressure difference between the two sides, a secondary flow is established which flows from the pressure side around the blade tip to the suction side. This secondary flow around the tip mixes with the primary main flow around the blade creating a vortex behind the blade tip. These tip vortices from each blade create further aerodynamic losses due to its high velocity and low pressures [14]. Also, a secondary flow is created around the root edge of the blade due to the pressure difference between the two sides of it. When this secondary flow mixes with the primary flow, a root vortex is created but aerodynamic losses created by root vortices are significantly lower as compared to the tip vortices.

The path lines of tip vortices from the blade tips are helical in nature in the downstream region. The distance between the tip vortices from each blade will be very small, forming a turbulent cylindrical shear layer [16]. Figure 3.6 illustrates the sketch of tip vortices from a three-blade rotor forming a cylindrical shear layer. This shear layer separates the wake flow from ambient flow.

As the vortices move further downstream, they are destabilized. There will be a turbulent mixing or diffusion process and the shear layer expands due to which the large-scale turbulent eddies break down into small-scale turbulent eddies [21]. The low velocity wake flow is mixed with the high velocity ambient flow due to the turbulence in the shear layer as shown in Figure 3.7 [27].

Figure 3.6: Cylindrical shear layer created by the tip vortices from a three-blade turbine

[18]

The region of mixing will expand inwards toward the centre of the wake as well as outwards which results in the expansion of the wake as shown in Figure 3.7. Thus, ABL plays an important role in the recovery of the velocity behind the turbine. Figure 3.8 shows the velocity profile in the wake viewed on the plane A-A (at hub height) from Figure 3.7 and Figure 3.9 shows the turbulence intensity profile on the horizontal plane A-A [27]. Due to the mixing, there will be a gradual recovery in the wake velocity and wake velocity profile gets flattened as moving downstream as shown in Figures 3.7 (vertically) and 3.8 (horizontally). At a particular distance in the downstream, shear layer will reach the centre of the wake. Up to this point of wake is the region called near wake region. In the near wake region, the effect of tip and root vortices are predominant. The wake flow in this region is largely influenced by the rotor geometry, number of blades, vortices, adherent or stalled flow etc. There are different values of x/D in the literature where the region of near wake terminates. According to Vermeer et al. [15] and Sanderse [21] the near wake ends at somewhere around $x/D=1$, but Gomez-Elivra et al. [16] suggest that the near wake region ends at some place between $x/D=2$ to 5 (where x is the distance downstream from the turbine (m) and D is the rotor diameter (m)).

Further away after the transition region (very short region and hence not indicated in Figure 3.7) is the far wake region. The rotor geometry has no influence on the far wake region. In this region, the wake is fully developed [16]. The effects in the far wake region are

experienced by the downstream turbines in the wind farm. While modelling the wind farm, the main focus will be on the far wake simulation [15].

Figure 3.7: Near wake region and far wake region behind the wind turbine [27]

Figure 3.8: Velocity profile of the wake along the horizontal plane at hub height [27]

Figure 3.9: Turbulence intensity profile of the wake along the horizontal plane at hub height [27]

The analysis and study of the near wake region is fairly complicated. There are a few assumptions to be made like steady, uniform and parallel flow. The complexity of this study arises when issues such as dynamic inflow, stall delay, shear layer formation and rotor tower interactions are taken into consideration.

The analysis of the far wake region is more important than the near wake region for the wind farm wake study as it is more focused on the impact when the turbines are placed in the wind farm in clusters. Study of near wake is however also needed as it gives initial conditions for the far wake flow.

Many experiments have been done on the wake analysis of the wind turbines. Due to the instabilities in the atmospheric boundary layer conditions, it is rather difficult to undergo measurements in the field and also to ensure repeatability. For this reason, turbines are modelled and the experiments are conducted in the wind tunnels. Some of the important wind tunnel experiments are conducted by Ross and Ainslie [28], Vermeulen and Builtjes [29], Hassan et al. [30], Smith [31], Smith and Taylor [32], Taylor and Smith [33] and Crespo and Hernandez [34].

The main advantages of wind tunnel experiments are the different alignment of wind turbines are considered, laminar or turbulent inflows, desired speed of the wind, and a number of reliable experiments can be done by varying the wind speed, number of revolutions etc. within a short period.

The major disadvantages of the wind tunnel experiments are difficulties in implementing real atmospheric conditions and the features of the dimensional analysis like scaling effects between the model and the real wind turbine.

Field experiments are necessary to understand the problems related to the realistic scenario. But it is difficult to control the experimental conditions and maintain uniformity. Some representative field experiments were conducted by Baker and Walker [35], Taylor et al. [36], Högström et al. [37], Frandsen et al. [38], Rados et al. [39].

The major issues with the field experiments are the changes in the direction of wind and the corresponding forces exerted. Recent work has developed numerical or virtual wake models which will be discussed in the following sections. In this work, CFD analysis of the NREL Phase VI Turbine was conducted by virtually placing the turbine with hub height 12m at different atmospheric conditions and their wake recovery and turbulence effects are analyzed and compared with the existing models. Since the experimental information about

the wake of NREL Phase VI Turbine was not available, the accuracy of this work is based on the theory discussed above and closure results to existing models. CFD modelling is described in Chapter 5 and results are discussed in Chapter 6.

3.3. Wake Models

3.3.1. Challenges and difficulties in wake modelling

During the past decades, the sizes of the wind turbines and wind farms have increased considerably. But there is a constraint regarding the space for the wind farm. Thus, it is necessary to analyze the positions of the maximum number of turbines in a given limited space to extract maximum possible power. Downstream turbines are subjected to velocity deficit inflow and increased loading. Hence the modelling of wind turbine spacing is necessary to reduce the load and maximize the power production.

Relatively accurate analysis and prediction of the large wind farms with all the effects of wakes can be computed by solving the full Navier-Stokes code with the aid of supercomputers. However, this is difficult and unrealistic as it requires more computational effort and time. Thus, it is necessary to simplify the governing equations that can provide faster numerical solutions but with realistic enough results.

Different wake models are developed for analyzing certain areas of the wakes because there are many factors that direct the wake flow in different areas. In spite of the development of numerous wake models and many experiments, it is difficult to understand the unsteady aerodynamic phenomena.

Modelling will be done separately for both the near wake and far wake regions as both exhibit different characteristics. Described in Section 3.3.2 is the overview of the various near wake and far wake models.

3.3.2. Near wake models

There is no formal definition for endpoint location of the near wake. It is the region behind the rotor where the properties of the rotor have influences on the wake. Downstream region less than two times of rotor diameter ($<2xD$) can be taken as near wake region [15, 16].

For the near wake models, it is necessary to analyze the properties of the aerodynamics of the rotor blade. Some of the most important and representative near wake models are:

- Blade Element Momentum method
- Vortex models
- Generalized Actuator models

3.3.2.1. Blade Element Momentum method

The BEM has been discussed in the previous chapter. It is a simple and most frequently used design tool for modelling wind turbines in industries. It assumes the air flows through annular segment and the lift and drag forces are calculated using momentum theory and aerofoil characteristics. The main disadvantage of the traditional BEM method is its simplicity. It does not consider the effects like unsteadiness, yaw misalignment and etc. Since the size of the wind turbines is increased in the past decade, the analysis of these effects is becoming more important. There are many correction factors which have been discussed in the previous chapter as well as to improve the accuracy of the BEM. In reality, inflow experienced by the wind turbine is not uniform but is unsteady both in velocity and direction. Also, the effect of stall delay plays an important role in unsteadiness. Hence, the assumption of inlet flow in BEM theory as steady and uniform is no more valid and realistic. Despite its limitation, BEM theory is frequently used in the industries because of its simplicity. Hence, there is an increased attempt to improve the analysis of BEM theory and one such attempt was done in Chapter 2 to improve the BEM for stall delay.

3.3.2.2. Vortex models

The vortex model was first suggested by Øye [40] and is according to the vortex theory. This method is more suited for the description of rotor aerodynamics and wake geometry and it compromises between steady BEM model and full rotor CFD analysis. This method was adopted for helicopter blade aerodynamics and their wake analysis by Landgrebe [41], Kocurek and Tangler [42] and Egolf and Landgrebe [43] and for wind turbine by Kocurek [44], Dumitrescu and Cardos [45] and Coton and Wang [46]. This model was based on the

following important assumptions of the vortex theory that the flow is incompressible ($\nabla \cdot \bar{u} = 0$), inviscid ($\mu = 0$) and irrotational ($\nabla \times \bar{u} = 0$) at all the points excluding the origin where the velocity will reach infinity [47]. In order to avoid singularity at the centre, the core rotates like a solid body. In this potential flow theory, vortices are used to represent the lift on the blade [48]. For irrotational flow, velocity potential is given by

$$\bar{u} = \nabla \Phi \quad (3.5)$$

Equation (3.6) represents Laplace equation and it is solved in this model.

$$\nabla^2 \Phi = 0 \quad (3.6)$$

where,

$\bar{u}$ is the velocity vector (m/s)

μ is fluid dynamic viscosity (Ns/m²)

Φ is velocity potential (m²/s)

There are three models as described in the literature:

- Lifting line method [45, 47, 49]
- Lifting surface method [46, 50]
- Vortex panel method [51-53]

These three models can be further classified based on the modelling of trailing vortex either as free wake [54-56] or prescribed wake [42, 44, 57]. Free wake model is computationally expensive, but prediction of wake geometry is better than the prescribed wake model. Experimental data are required for the prescribed wake model. As discussed in Chapter 2, Sant et al. [58] and Schepers and van Rooij [59] have used free wake model for NREL Phase VI Turbine rotor analysis and Tangler and Kocurek [60] and Breton et al. [61] have used prescribed wake model for the NREL Phase VI Turbine. This method is not detailed in this work since the focus is on improving BEM analysis for unsteady conditions and coupling with CFD.

3.3.2.3. Generalized actuator models

The generalized actuator models are based on the computation of momentum equations and extension of BEM analysis. In this model, force 'f' computed using analytical methods like momentum theory or blade element theory or combination of both explicitly is applied to the momentum equation (Equation (3.7)). The Computational Fluid Dynamics (CFD) is used to solve the flow around the turbine. Based on the method of computation of force and their implementation, these models can be called as Actuator disk, Actuator line and Actuator surface models.

$$\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla \vec{u} = \vec{f} - \frac{1}{\rho} \nabla p + \nu \nabla^2 \vec{u} \quad (3.7)$$

where,

$\vec{u}$ is velocity vector (m/s)

p is pressure (Pa)

ρ is density (kg/m³)

ν is kinematic viscosity (m²/s)

$\vec{f}$ is force vector (N)

Wu [62], Hough and Ordway [63], Greenberg [64], Sørensen and Myken [65], Conway [66], Sørensen and Kock [67], Madsen [68] and Conway [69] have undergone research in the generalised actuator disk models. In the actuator disk model, turbine is replaced with a disk. Sørensen and his colleagues [70-72] proposed a new approach called actuator line model. In this approach, blades are replaced with lines and forces computed along a blade span are applied to the line. Further extension of the actuator line model is the actuator surface model. In this model, the blades are replaced by a thin surface and forces are distributed chordwise. Shen et al. [73, 74] and Dobrev et al. [75] have implemented this method in their corresponding works. It has fewer limitations compared to BEM analysis.

3.3.2.4. Navier-Stokes models

Computation of the full Navier-Stokes equations around the wind turbine in the domain can give reasonable results about the flow in the rotor and the near wake region. But in realistic high computer power is required. A review is done on the usage of CFD code by Sanderse [21].

3.3.2.4.1. Reynolds Averaged Navier-Stokes (RANS)

In order to solve the turbulent Navier-Stokes equations, the flow properties are divided into mean part and fluctuating part [76, 77]. For example, in Equation (3.8) the flow property velocity (u_i) is split into an average term ($\bar{u}_i$) and a fluctuation term (u'_i).

$$u_i = \bar{u}_i + u'_i \quad (3.8)$$

where,

u_i is the instantaneous velocity of the turbulent flow in the i^{th} direction (m/s)

$\bar{u}_i$ is the mean velocity (time-averaged) component in the i^{th} direction over a time interval T (m/s)

u'_i is the fluctuation velocity component in the i^{th} direction (m/s)

On substituting this into the general momentum and continuity equations and by time averaging, it results in RANS (Reynolds Averaged Navier-Stokes) equations. This term is same as the conventional Navier-Stokes equations with an additional term called Reynolds stress tensor (τ'_{ij}).

$$\tau'_{ij} = -\rho \overline{u'_i u'_j} \quad (3.9)$$

An additional expression is needed in order to get the closure solution. The turbulence models are used as an additional expression to arrive at this closure. These are relationship between the Reynolds stress and average flow field. The most commonly used turbulent model is the $k-\varepsilon$ model. This method will be discussed briefly in Chapter 4.

3.3.2.4.2. Large Eddy Simulations (LES)

This method is better than the RANS method as it depends comparatively less on the turbulence models. Wind turbines usually operate at large Reynolds number in the order of 1×10^6 . Thus, the difference between the largest length scale and the smallest length scale are significant. This LES method computes the larger eddies that are greater than grid size and the small eddies are modelled. It is gaining much attention in wind turbine research and modelling as this method can easily compute and model the velocity deficit, turbulence intensities, flow separation and momentum losses etc [78-80, 76]. This method is also discussed in Chapter 4.

3.3.3. Far wake models

In general, far wake regions commence at a distance larger than two times of rotor diameter ($2xD$) from the plane of the blades [29, 30, 34]. In this region, the wake is fully developed. Moreover, the aerodynamic properties of the rotor have no influence in this region. Therefore, this region is relatively simple to model as compared to the near wake region. Usually in the wake models developed, the near wake and far wake regions are distinctly separated but in actual situation there exist no distinct separation between the two regions. Far wake models can be divided into categories

- Kinematic models (also called explicit models)
- Field models (also called implicit models)

3.3.3.1. Kinematic models

These models are simple analytical models and require less effort for computation. Hence, these models are well suited for the prediction of the deficits in velocity profile of large wind farms. But the disadvantage of this model is that they have to be coupled with separate turbulence models in order to describe the turbulence intensity created in the wake. Most popular kinematic models are

- Jensen model
- Larsen model

- Frandsen model
- Ishihara model

3.2.3.1.1. Jensen model (PARK model)

This is the simplest and oldest model. It was developed by Jensen [81] and is described in the paper by Katic et al. [82] and in the technical report by Thørgersen et al. [83]. In this model, wake is assumed to be linearly expanding with velocity deficit only dependent on the distance downstream of the rotor as shown in Figure 3.10. The velocity is constant in the radial direction of the wake. The velocity profile will be hat shaped as shown in Figure 3.10. This model includes rotor diameter and the coefficient of thrust to predict the velocity profile.

The wake diameter (D_w) is given by

$$D_w = D + 2k_w x \quad (3.10)$$

where,

D is rotor diameter (m)

x is distance downstream of the turbine (m)

k_w is wake decay constant

Relative distance downstream behind the turbine is given by

$$s = \frac{x}{D} \quad (3.11)$$

Hence Equation (3.10) becomes

$$D_w = D(1 + 2k_w s) \quad (3.12)$$

Figure 3.10: Jensen wake model [82]

Velocity in the (fully developed) wake is given by

$$U_w = U_\infty \left[1 - \left(\frac{1 - \sqrt{(1 - C_T)}}{(1 + 2k_w s)^2} \right) \right] \quad (3.13)$$

where,

C_T is thrust coefficient of the rotor

U_∞ is freestream velocity (m/s)

U_{w0} is the velocity of wake at $x=0$ (m/s) (shown in Figure 3.10)

The value of k_w depends on various factors like atmospheric stability, turbulence in the atmosphere, wake induced turbulence etc. The standard decay constants recommended in the WAsP (Wind Atlas Analysis and Application Program which is PC program developed and distributed by the Department of Wind Energy at the Technical University of Denmark, Denmark) are 0.075 for onshore and 0.05 for offshore [84]. But in general, Equation (3.14) provides wake decay constant k_w for different terrains based on roughness length (z_o) and hub height (z_{hub}) [85].

$$k_w = \frac{0.5}{\ln \left(\frac{z_{hub}}{z_o} \right)} \quad (3.14)$$

3.3.3.1.2. Larsen model

This model was developed by Larsen in 1988 [86] and further improved by Larsen et al. [87] in 1996 and is described in the technical report of Thørgersen et al. [83]. This model is also known as an EWTSII model (European Wind Turbine Standard II). The model is based on the Prandtl's Turbulent Boundary Layer equations and mixing length theory is used. The model describes a Gaussian shape velocity deficit profile. The following assumptions on wake are made in this model

- Incompressible
- Stationary
- Axisymmetry

The wake velocity downstream of the turbine is the function of both downstream axial position (x) and radial position (r) is given by

$$U_w = U_\infty - \frac{U_\infty}{9} [C_T A (x + x_o)^{-2}]^{\frac{1}{3}} \left(r^{\frac{3}{2}} [3c_1^2 C_T A (x + x_o)]^{\frac{1}{2}} - \left(\frac{35}{2\pi} \right)^{\frac{3}{10}} (3c_1^2)^{-\frac{1}{5}} \right)^2 \quad (3.15)$$

Wake diameter is given by

$$D_w = 2 \left(\frac{35}{2\pi} \right)^{\frac{1}{5}} (3c_1^2)^{\frac{1}{5}} [C_T A (x + x_o)]^{\frac{1}{3}} \quad (3.16)$$

where,

C_T is thrust coefficient

A is the rotor swept area (m^2)

x is distance downstream of the turbine (m)

x_o is rotor position of applied coordinate system (m)

c_1 is non-dimensional mixing length

These two constants are estimated using the expressions [88] as follows,

$$x_o = \frac{9.5D}{\left(\frac{2R_{9.5}}{D_{eff}} \right)^3 - 1} \quad (3.17)$$

$$c_1 = \left(\frac{D_{\text{eff}}}{2} \right)^{\frac{5}{2}} \left(\frac{105}{2\pi} \right)^{-\frac{1}{2}} (C_T A_{x_o})^{-\frac{5}{6}} \quad (3.18)$$

where,

D_{eff} is the effective rotor diameter in (m) and is given by

$$D_{\text{eff}} = D \sqrt{\frac{1 + \sqrt{1 - C_T}}{2\sqrt{1 - C_T}}} \quad (3.19)$$

$R_{9.5}$ is the radius of the wake at 9.5 times rotor diameter behind the turbine (m).

This model also considers the blockage effect induced by the ground if the wake exceeds the hub diameter as it expands downstream and starts interacting with the ground. $R_{9.5}$ is given by

$$R_{9.5} = 0.5[R_{\text{nb}} + \min(z_{\text{hub}}, R_{\text{nb}})] \quad (3.20)$$

where,

R_{nb} is an empirical relation in association with ambient turbulence intensity (I_a) at the hub height.

$$R_{\text{nb}} = \max(1.08D, 1.08D + 21.7D(I_a - 0.05)) \quad (3.21)$$

z_{hub} is the hub height in (m)

3.3.3.1.3. Frandsen model

This model was developed by Frandsen et al. [89] and was presented at the European Wind Energy Conference (EWEC) in 2006. The model is similar to Jensen model and it also considers hat shaped velocity deficit profile, but this model has good prediction for neighbour turbine interactions and multiple wakes [89]. In this model, wakes are modelled based on three regimes. Highlights of these regimes are described as follows.

- (i) First regime
 - Single row of turbines
 - Multiple wakes but no interaction with each other
- (ii) Second regime

- Wakes meet and interact with each other
- (iii) Third regime
- Wake flow is balanced with ABL

In the present work, since the wake behind a single turbine is considered, only expressions related to the first regime and single wake are described as follows.

Wake diameter is given by

$$D_w = D \left(\beta_1^{\frac{k_1}{2}} + \alpha_1 s \right)^{\frac{1}{k_1}} \quad (3.22)$$

where,

$$\beta_1 = \frac{1 + \sqrt{1 - C_T}}{2\sqrt{1 - C_T}} = \left(\frac{D_{eff}}{D} \right)^2 \quad (3.23)$$

$$s = \frac{x}{D} \quad (3.24)$$

where,

D is rotor diameter (m)

x is distance downstream of the turbine (m)

D_{eff} is the effective rotor diameter in (m)

β_1 is a parameter representing wake expansion

α_1 is constant (=0.5 [90])

C_T is thrust coefficient

s is the relative distance downstream behind the turbine

k_1 is 3 (for Schlichting solution for wake expansion [89, 90]) or 2 (for Square root shape [89])

Wake velocity is given by

$$U_w = \frac{U_\infty}{2} \left(1 \pm \sqrt{1 - 2 \frac{A}{A_{wake}} C_T} \right) \quad (3.25)$$

where,

A is the rotor swept area (m²)

A_{wake} is wake area in (m²) (based on wake radius)

‘+’ is used inside the square root in the expression (3.25) if the axial induction factor a ≤ 0.5

and ‘-’ is used if a > 0.5

3.3.3.1.4. Ishihara model

This is an analytical model developed by Ishihara et al. [91] based on the wind tunnel analysis of scaled Mitsubishi turbine. The advantage of this method is that it includes the effect of ABL turbulence and turbulence generated by the rotor in the wake. Velocity profile is Gaussian and wake velocity is given by

$$U_w = U_\infty - \frac{\sqrt{C_T} U_\infty}{32} \left(\frac{1.666}{0.27} \right)^2 \left(\frac{x}{D} \right)^{-p} \exp \left(-\frac{r_w^2}{b^2} \right) \quad (3.26)$$

where,

U_w is wake velocity (m/s)

C_T is thrust coefficient of the rotor

D is rotor diameter (m)

x is distance downstream of the turbine (m)

U_∞ is freestream velocity (m/s)

r_w is wake local radius (m)

b is wake width (m)

$$b = \frac{0.27 C_T^{\frac{1}{4}}}{0.833} D \left(1 - \frac{p_w}{2} \right) x^{\frac{p_w}{2}} \quad (3.27)$$

p_w is a parameter and is a function of ambient and wake turbulence intensities

$$p_w = 6(I_a + I_{wake}) \quad (3.28)$$

where,

I_a is ambient turbulence intensity

I_{wake} is rotor generated wake turbulent intensity

$$I_{\text{wake}} = \frac{0.004C_T}{\max(I_a, 0.03)} \left(1 - \exp \left[-4 \left(\frac{x}{10D} \right)^2 \right] \right) \quad (3.29)$$

Table 3.2 summarises the input parameters required for modelling for the wake models using kinematic methods (analytical).

Parameters	Jensen model [81]	Larsen model [86,87]	Frandsen model [89]	Ishihara model [91]
Freestream input velocity (u_∞) in m/s	✓	✓	✓	✓
Distance downstream of the turbine(x) in m	✓	✓	✓	✓
Thrust coefficient (C_T)	✓	✓	✓	✓
Radius from hub (r) in m		✓		✓
Rotor diameter(D) in m	✓	✓	✓	✓
Hub height (z_{hub}) in m	✓	✓		
Turbulence intensity (I)		✓		✓

Table 3.2: Summary of input parameters for different kinematic models

3.3.3.2. Field models

Field models are based on solving the RANS equations to obtain flow field in the wind farm.

This model is more advanced than kinematic modelling [92], but it is complicated as it requires many numerical techniques. The best-known field models are as follows

- Ainslie model
- UPMWAKE model

3.3.3.2.1. Ainslie model

This model [93, 94, 83] is a well-known 2D model. It assumes axial symmetric flow and uses eddy viscosity for closure of time-averaged Navier-Stokes equation. Figure 3.11 shows the graphical flow representation of this model [83].

Figure 3.11: Flow behind the turbine based on Ainslie model [83]

This model uses cylindrical coordinates for the continuity equation and Navier-Stokes equation. They are given by

$$\frac{1}{r} \frac{\partial(rv)}{\partial r} + \frac{\partial u}{\partial x} = 0 \quad (3.30)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial r} = -\frac{1}{r} \frac{\partial(r \overline{u'v'})}{\partial r} \quad (3.31)$$

Eddy viscosity ($\varepsilon(x)$) is used to model the Reynolds stress ($-\overline{u'v'}$) and the relationship is given as Equation (3.32)

$$-\overline{u'v'} = \varepsilon(x) \frac{\partial u}{\partial r} \quad (3.32)$$

Eddy viscosity is again divided into two terms as follows

$$\varepsilon(x) = \varepsilon_{ambient}(x) + \varepsilon_{wake}(x) \quad (3.33)$$

$$\varepsilon_{wake}(x) = 0.015b(u_{\infty} - u_c(x)) \quad (3.34)$$

where,

b is the wake width in (m)

u_{∞} is the freestream velocity in (m/s)

x is distance downstream of the turbine (m)

$u_c(x)$ is velocity at centre line of wake at the distance x in (m/s)

In this model pressure gradient in the wake are neglected and this assumption is not valid in the near wake region. Hence this model cannot be used in the near wake region and has to be initialised with Gaussian wind velocity wake profile after the near wake region (2D) and is given by

$$1 - \frac{u}{u_{\infty}} = (u_{\infty} - u_c) \exp \left(-3.56 \left(\frac{r}{b} \right)^2 \right) \quad (3.35)$$

where,

$(u_{\infty} - u_c)$ in Equation (3.35) is initial velocity deficit and is given by

$$(u_{\infty} - u_c) = C_T - 0.05 - (16C_T - 0.5) \frac{I_a}{1000} \quad (3.36)$$

where,

I_a is ambient turbulent intensity in (%)

Wake width b is defined as the radial distance at which velocity deficit is 2.83% of u_c and is given by

$$b = \sqrt{\frac{3.56C_T}{4(u_{\infty} - u_c)(2 - (u_{\infty} - u_c))}} \quad (3.37)$$

3.3.3.2.2. UPMWAKE model

This model was developed by Crespo et al. [95] and Crespo and Hernández [96] in Universidad Polytechnica de Madrid and is described in the work of Prospathopoulos et al. [92]. The code is based on the 3D parabolised RANS equation and it uses k-epsilon turbulence model for closure solution. This model does not assume axisymmetric and it accounts for the effect of neutral atmospheric boundary layer. ECN (Energy Research Center of the Netherlands) has used this code in its wind farm program and named it as WAKEFORM [92]. In the original version, the turbine is replaced with a permeable disk and near wake region is not properly modelled due to the effect of parabolisation and also the axial pressure gradient is not taken into account. Original version used hat shaped velocity deficit and far wake is initialized at a distance of 2.25 times rotor diameter downstream.

The Gaussian wind velocity deficit was later applied to initialise the near wake region. The free wake vortex model is used to improve the code to account for near wake analysis by considering the pressure gradient in the improved version of this model by ECN [97].

3.4. Turbulence Intensity Modelling

It is necessary to model the turbulence level in the wake as downstream wind turbine experiencing high turbulence might result in failure as described in Section 3.2. Turbulence intensity is used to define the turbulence level both in the ambient and wake region. The turbulence intensity (I) is defined as the ratio of standard deviation of the variation in the wind speed to the mean wind speed [6].

$$I = \frac{\sigma_u}{\bar{u}} \quad (3.38)$$

where,

σ_u is the standard deviation of wind speed variations about the mean wind speed $\bar{u}$ (m/s)

Turbulence Intensity value only depends on the surface roughness length in the neutral atmospheric boundary layer. Different terrains have different roughness length value [6, 92]. The relationship between turbulence intensity and surface roughness [6, 83] is based on Equation (3.39).

$$I = A_1 K \frac{1}{\ln\left(\frac{z}{z_o}\right)} \quad (3.39)$$

where,

A_1 is constant and value is varying from 2.5 to 1.8 [83]

K is von Karman constant

Product of A_1 and K is approximated to be 1. Then the following expression is obtained.

$$I \approx \frac{1}{\ln\left(\frac{z}{z_o}\right)} \quad (3.40)$$

where,

z is the height above the ground (m)

z_0 is the roughness length of the terrain (m)

Equation (3.41) provides the relationship between turbulence intensity and turbulence kinetic energy [98, 99]. This equation is more convenient to use especially during the post processing of the CFD analysis to determine the turbulence intensity.

$$I = \frac{\sqrt{\frac{2}{3}k}}{\bar{u}(z)} \quad (3.41)$$

where k is turbulence kinetic energy in J/kg or m^2/s^2

As discussed in Section 3.2, turbulence in the wake will higher than the ambient due to the turbulence added by the rotor. Total turbulence intensity in the wake can be expressed in terms of atmospheric inflow turbulence intensity and turbulence intensity added by the rotor.

$$I_{wake} = (I_a^2 + I_+^2)^{0.5} \quad (3.42)$$

I_{wake} is the total wake turbulence intensity

I_+ is the added turbulence intensity

I_a is the ambient turbulence intensity

There are several empirical relations proposed to find the added turbulence intensity in the wake region.

Quarton and Ainslie [100] proposed an empirical relation of added turbulence intensity as a function of ambient turbulence intensity (I_a), thrust coefficient (C_T) and downstream distance (x) as follows

$$I_+ = 4.8C_T^{0.7}I_a^{0.68}\left(\frac{x}{x_n}\right)^{-0.57} \quad (3.43)$$

where,

x_n is the distance of the near wake region. Generally, it is assumed to be 2 times downstream distance of the turbine. But Vermeulen [101] suggested Expression (3.44) for x_n to have a better prediction of the near wake distance.

$$x_n = \frac{\sqrt{(0.214 + 0.144m_1)}(1 - \sqrt{(0.134 + 0.124m_1)})}{(1 - \sqrt{(0.214 + 0.144m_1)})\sqrt{(0.134 + 0.124m_1)}} \left(\frac{dr}{dx} \right) r_o \quad (3.44)$$

where,

m_1 is a parameter and is given by

$$m_1 = \frac{1}{\sqrt{1 - C_T}} \quad (3.45)$$

r_o is a parameter and is given by

$$r_o = R \sqrt{\frac{m_1 + 1}{2}} \quad (3.46)$$

where,

R is radius of the rotor in (m)

dr/dx is the wake growth and is given by

$$\frac{dr}{dx} = \sqrt{\left(\left(\frac{dr}{dx} \right)_{\text{ambient}}^2 + \left(\frac{dr}{dx} \right)_{\text{sheargenerated}}^2 + \left(\frac{dr}{dx} \right)_{\text{mechanical}}^2 \right)} \quad (3.47)$$

Contribution of ambient turbulence is given by

$$\left(\frac{dr}{dx} \right)_{\text{ambient}} = 2.5I_a + 0.005 \quad (3.48)$$

Contribution of shear generated turbulence is given by

$$\left(\frac{dr}{dx} \right)_{\text{sheargenerated}} = \frac{(1 - m_1)\sqrt{1.49 + m_1}}{9.76(1 + m_1)} \quad (3.49)$$

Contribution of rotor generated turbulence is given by

$$\left(\frac{dr}{dx} \right)_{\text{mechanical}} = 0.012B\lambda \quad (3.50)$$

B is number of blades and λ is the tip speed ratio of the rotor.

Hassan [102] improved the relation provided by Quarton and Ainslie [100] and proposed the following expression

$$I_+ = 5.7 C_T^{0.7} I_a^{0.68} \left(\frac{x}{x_n} \right)^{-0.96} \quad (3.51)$$

The near wake length is calculated using the same equations from (3.44) to (3.50).

Larsen et al. [87] proposed the Expression (3.52) for added turbulence and is described in the technical report of Thørgersen et al. [83].

$$I_+ = 0.29 s^{\frac{-1}{3}} \sqrt{\left(1 - \left(\sqrt{1 - C_T}\right)\right)} \quad (3.52)$$

where,

$$s = \frac{x}{D} \quad (3.53)$$

The model proposed by Frandsen and Thørgersen [103] takes the wind farm layout in the consideration.

$$I_+ = 0.5 \left(I_a + \sqrt{I_a^2 + I_{++}^2} \right) \quad (3.54)$$

where,

$$I_{++} = \frac{0.36}{1 + 0.2 \sqrt{s_1 s_2 / C_T}} \quad (3.55)$$

where,

I_{++} is added turbulence above the hub height due to additional surface roughness from the turbines in the wind farm [6]

s_1 and s_2 are the inter-turbine spacings (normalized by the rotor diameter) within a row and between the rows respectively. In the present study, this model is not being considered since only wakes are analyzed for wake behind the single turbine. Table 3.3 shows the input parameters necessary for the turbulence intensity models.

Parameters	Quarton and Ainslie model [100]	Hassan model [102]	Larsen model [87]
Distance downstream of the turbine(x) in m	✓	✓	✓
Thrust coefficient C_T	✓	✓	✓
Near wake length in m	✓	✓	
Ambient turbulence intensity (I_a)	✓	✓	

Table 3.3: Summary of input parameters for different wake turbulence intensity models

3.5. Hybrid models

These models are based on associating different wake models for different regions to eliminate the limitations of the conventional wake model [38]. The various models are combined in an effective way to model the flow field from the rotor to the far-field.

Voutsinas et al. [104, 105] have developed the hybrid wake model. The computational domain is segmented into three regions and three different models are used for resolving the flow fields. The lifting line code is used to model the rotor. Then viscous RANS code with $k-\varepsilon$ turbulence model is used to model the flow field up to 3 to 4 times of rotor diameter downstream of the turbine. Next, the output is used to calculate the velocity deficit profile in the far-field region. In the far-field region, a self-similar velocity profile like kinematic models is used.

A wide range of research has been conducted and is still in progress to understand the flow field behind the wind turbines in the wind farm. Even with the advancement of high speed computing, a full CFD simulation of the flow fields in the wind farm remains time consuming. From the industrial point of view, there should be a compromise between the accuracy and the computing time. Thus, this opens the way for research to expand the BEM theory to include unsteady 3D effects. Another promising area of the research is analysis of hybrid models in which different suitable models are applied to different regions.

CHAPTER 4: GOVERNING EQUATIONS AND NUMERICAL METHODS

4.1. Governing Equations

The governing equations of the fluid motions are obtained from the conservation of mass, momentum and energy. Equation (4.2) provides Navier-Stokes Equation in tensor notation.

Continuity Equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad (4.1)$$

Momentum Equation or Navier-Stokes Equation

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j + p \delta_{ij} - \tau_{ij}) = \rho f'_i \quad (4.2)$$

where,

u_i is the velocity component in the i^{th} direction (m/s)

p is the pressure (Pa)

ρ is the density (kg/m³)

τ_{ij} is the normal and shear stresses acting on the surfaces of the fluid particles (N/m²)

f'_i is body force per unit mass in the i^{th} direction (N/kg)

δ_{ij} is the Kronecker delta

i and j take the values 1,2 and 3

In this CFD study, energy equation is not considered because thermal phenomenon is taken as negligible.

For incompressible flow, density (ρ) is constant.

$$\frac{\partial \rho}{\partial t} = 0 \text{ with constant value of } \rho.$$

4.2. Reynolds Averaging

Turbulent flow is characterized by its irregular and chaotic motion. The flow properties can be decomposed into mean and fluctuation components like

$$u_i = \bar{u}_i + u'_i \quad (4.3)$$

where,

u_i is the instantaneous velocity of the turbulent flow in the i^{th} direction (m/s)

$\bar{u}_i$ is the mean velocity (time-averaged) component in the i^{th} direction over a time interval T' (m/s).

$$\bar{u}_i = \lim_{T' \rightarrow \infty} \frac{1}{T'} \int_0^{T'} u_i(t) dt \quad (4.4)$$

u'_i is the fluctuation velocity component in the i^{th} direction (m/s)

p is the instantaneous pressure of the turbulent flow (Pa)

$\bar{p}$ is the mean component of the pressure term (Pa)

$$\bar{p} = \lim_{T' \rightarrow \infty} \frac{1}{T'} \int_0^{T'} p(t) dt \quad (4.5)$$

p' is the fluctuation component of the pressure term (Pa)

Substituting $\frac{\partial \rho}{\partial t} = 0$ (incompressible flow with a constant density value) and the Equation

(4.3) into Equations (4.1) and (4.2) and taking time averaging and further manipulation yields

$$\frac{\partial(\bar{u}_i)}{\partial x_i} = 0 \quad (4.6)$$

$$\frac{\partial}{\partial t}(\rho \bar{u}_i) + \rho \frac{\partial \bar{u}_j \bar{u}_i}{\partial x_j} = \rho \bar{f}_i' + \frac{\partial}{\partial x_j} (-\bar{p} \delta_{ij} + \bar{\tau}_{ij} - \rho \overline{u'_i u'_j}) \quad (4.7)$$

where,

ρ is the density (kg/m³)

$\bar{u}_i$ is the mean velocity (time-averaged) component in the i^{th} direction over a time interval T' (m/s).

$\bar{f}_i'$ is the mean component of the body force per unit mass in the i^{th} direction (N/kg)

$\bar{\tau}_{ij}$ is the mean component of the stresses acting on the surfaces of the fluid particles (N/m^2)

This is the Reynolds Averaged Navier-Stokes Equation (RANS) and the term $-\rho \overline{u'_i u'_j}$ is named as Reynolds or turbulence stress tensor.

There are four equations for the mean flow (one continuity equation and three momentum equations). But there are ten unknowns ($\bar{u}_i, \bar{p}$ and six components of Reynolds stress tensor $-\rho \overline{u'_i u'_j}$). The system is unclosed. This is termed as closure problem of turbulence [1].

4.3. Turbulence Modelling (TM)

Turbulence modelling provides necessary closure to RANS equation by representing the unknown (Reynolds stress) in terms of the known values (mean flow) [1]. Types of closure models are:

- Algebraic (zero-equation) models
 - Mixing length [1]
 - Cebeci-Smith Model [1]
 - Baldwin-Lomax Model [1]
- One-equation models
 - Baldwin-Barth model [1]
 - Spalart-Allmaras model [1]
- Two-equation models
 - k - ε model [1]
 - k - ω model [1]
 - k - ω SST model [1]
- Four-equation (v^2 - f) models (where v^2 represents velocity scale and f represents elliptical relaxation factor) [2]
- Reynolds-stress (seven-equation) models [3]

k- ε model was the most commonly used model, but it over-predicts value of the turbulence length scale in the near wall region and hence results in higher shear stress value. To overcome this problem, Wilcox [1] developed the k- ω turbulence closure model, which is good to model the free shear flows, adverse pressure gradient flows and separated flows. k- ω SST model which is explained in Section 4.3.1 was initially chosen for CFD analysis of NREL Phase VI Turbine as it is good in determining the wall effects which is necessary for the computation of $C_{L,3D}$ and $C_{D,3D}$ for improving BEM analysis (discussed in Chapter 2). In Chapter 3, it can be noticed that boundary layer of ABL is modelled using k- ε model. Since the main objective of analysis of NREL Phase VI Turbine in different ABL is to study the effect of wake which is away from the wall (blades), k- ε model is used for the simulation in ABL.

4.3.1. k- ω SST model

To capture the qualities of the k- ω model in the near-wall region and the qualities of the k- ε in the far-field simultaneously, k- ω SST (Shear Stress Transport) model was proposed by Menter [4] in 1994. This model takes advantage of both k- ω and k- ε models. Both the models k- ω and k- ε are multiplied by blending functions and are added together. It changes from k- ω model in the near wall to k- ε model in the outer region in providing better treatment for boundary layer. This model includes a damped cross-diffusion derivative term in the transport equation of ω (Equation (4.9)). This model is reliable and predict more accurately for flow with adverse pressure gradient, wings etc than the standard k- ω model.

The transport equations for this k- ω SST model are

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\Gamma_k \frac{\partial k}{\partial x_j} \right] + \tilde{G}_k - Y_k + S_k \quad (4.8)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left[\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + D_\omega + S_\omega \quad (4.9)$$

where,

ρ is the density (kg/m³)

k is the turbulence kinetic energy which determines the energy in turbulence (m^2/s^2)

ω is the specific dissipation rate of k ($1/\text{s}$)

u_i is the component of velocity in the i^{th} direction (m/s)

$\tilde{G}_k$ is the turbulence kinetic energy generation by mean velocity gradients ($\text{kg}/(\text{ms}^2)$)

G_ω represents the specific dissipation rate generation ($\text{kg}/(\text{m}^2\text{s}^2)$)

Y_k is the dissipation of k ($\text{kg}/(\text{ms}^2)$)

Y_ω is the dissipation of ω ($\text{kg}/(\text{m}^2\text{s}^2)$)

Γ_k is the effective diffusivity of k (Ns/m^2)

Γ_ω is the effective diffusivity of ω (Ns/m^2)

D_ω is the cross-diffusion term ($\text{kg}/(\text{m}^2\text{s}^2)$)

S_k is the user-defined source terms for k ($\text{kg}/(\text{ms}^2)$)

S_ω is the user-defined source terms for ω ($\text{kg}/(\text{m}^2\text{s}^2)$)

The term for turbulence kinetic energy generation $\tilde{G}_k$ is given by

$$\tilde{G}_k = \min(G_k, 10 \rho \beta^* k \omega) \quad (4.10)$$

The term for production of ω , G_ω is given by

$$G_\omega = \frac{\alpha_2}{v_t} G_k \quad (4.11)$$

where,

$$G_k = -\rho \overline{u'_i u'_j} \frac{\partial u_j}{\partial x_i} \quad (4.12)$$

$-\rho \overline{u'_i u'_j}$ is the Reynolds stress tensor

v_t is the kinematic turbulent viscosity which is given by

$$v_t = \frac{\mu_t}{\rho} \quad (4.13)$$

μ_t is the turbulent viscosity for this model is computed as

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max\left[\frac{1}{\alpha^*}, \frac{S F_2}{2}\right]} \quad (4.14)$$

where,

S is the modulus of the mean rate of strain tensor S_{ij} and is given by

$$S \equiv \sqrt{(2S_{ij}S_{ij})} \quad (4.15)$$

The coefficient α_2 is given by

$$\alpha_2 = \frac{\alpha_\infty}{\alpha^*} \left(\frac{\left(\frac{1}{9}\right) + \left(\frac{(\rho k/\mu \omega)}{2.95}\right)}{1 + \left(\frac{(\rho k/\mu \omega)}{2.95}\right)} \right) \quad (4.16)$$

$$\alpha_\infty = F_1 \alpha_{\infty,1} + (1 - F_1) \alpha_{\infty,2} \quad (4.17)$$

$$\begin{aligned} \alpha_{\infty,1} &= \frac{0.075}{0.09} - \frac{0.41^2}{2 * \sqrt{0.09}} \\ &= 0.5531 \end{aligned} \quad (4.18)$$

$$\begin{aligned} \alpha_{\infty,2} &= \frac{0.0828}{0.09} - \frac{0.41^2}{1.168 * \sqrt{0.09}} \\ &= 0.4402 \end{aligned} \quad (4.19)$$

$$\alpha^* = \alpha_\infty^* \left(\frac{\left(\frac{\beta_i}{3}\right) + \left(\frac{(\rho k/\mu \omega)}{6}\right)}{1 + \left(\frac{(\rho k/\mu \omega)}{6}\right)} \right) \quad (4.20)$$

$$\beta_i = 0.075F_1 + 0.0828(1 - F_1) \quad (4.21)$$

$$\beta^* = \beta_i^* [1 + 1.5 F(M_t)] \quad (4.22)$$

$$\beta_i^* = 0.09 \left(\frac{\frac{4}{15} + \left(\frac{(\rho k/\mu \omega)}{8}\right)^4}{1 + \left(\frac{(\rho k/\mu \omega)}{8}\right)^4} \right) \quad (4.23)$$

$$F(M_t) = \begin{cases} 0 & \text{for } \left(\frac{2k}{\gamma \bar{R} T_e} \right)^{0.5} \leq 0.25 \\ \left(\frac{2k}{\gamma \bar{R} T_e} \right) - (0.25)^2 & \text{for } \left(\frac{2k}{\gamma \bar{R} T_e} \right)^{0.5} > 0.25 \end{cases} \quad (4.24)$$

where,

$\bar{R}$ is Gas law constant

The blending functions F_1 and F_2 are given by

$$F_1 = \tanh(\phi_{F1}^4) \quad (4.25)$$

$$\phi_{F1} = \min \left[\max \left(\frac{\sqrt{k}}{0.09\omega y_1}, \frac{500\mu}{\rho y_1^2 \omega} \right), \frac{4\rho k}{1.168^+ D_\omega^+ y_1^2} \right] \quad (4.26)$$

$$D_\omega^+ = \max \left[2\rho \frac{1}{1.168} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10} \right] \quad (4.27)$$

$$F_2 = \tanh(\phi_{F2}^2) \quad (4.28)$$

$$\phi_{F2} = \max \left[2 \frac{\sqrt{k}}{0.09\omega y_1}, \frac{500\mu}{\rho y_1^2 \omega} \right] \quad (4.29)$$

where,

μ is the fluid dynamic viscosity (Ns/m²)

T_e is temperature (K)

γ is the ratio of specific heats (specific heat capacity at constant pressure/specific heat capacity at constant volume)

y_1 is the coordinate normal to the wall (m)

D_ω^+ is the positive portion in the cross-diffusion term D_ω

The dissipation of k and ω are given by

$$Y_k = \rho \beta^* k \omega \quad (4.30)$$

$$Y_\omega = \rho \beta k \omega^2 \quad (4.31)$$

where,

$$\beta = \beta_i \left[1 - 1.5 \left(\frac{\beta_i^*}{\beta_i} \right) F(M_t) \right] \quad (4.32)$$

The effective diffusivities is given by

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \quad (4.33)$$

$$\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \quad (4.34)$$

where,

σ_k is the turbulent Prandtl numbers for k

σ_ω is the turbulent Prandtl numbers for ω

$$\sigma_k = \frac{1}{\frac{F_1}{1.176} + \frac{1-F_1}{1.0}} \quad (4.35)$$

$$\sigma_\omega = \frac{1}{\frac{F_1}{2.0} + \frac{1-F_1}{1.168}} \quad (4.36)$$

The cross-diffusion term is given by

$$D_\omega = 2 * 1.168 * (1 - F_1) \rho \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (4.37)$$

More detailed explanation of the transport equations can be found in the reference [5].

4.3.2. Standard k-ε model

The transport equations of k-ε model are given by

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (4.38)$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (4.39)$$

where,

ρ is the density (kg/m³)

k is the turbulence kinetic energy which determines the energy in turbulence (m²/s²)

ε is the turbulent dissipation rate of k (m^2/s^3)

u_i is the component of velocity in the i^{th} direction (m/s)

G_k is the turbulence kinetic energy generation by mean velocity gradients ($kg/(ms^2)$)

G_b is the turbulence kinetic energy generation by buoyancy ($kg/(ms^2)$)

Y_M is the dilation diffusion term ($kg/(ms^2)$)

σ_k and σ_ε is the turbulent Prandtl numbers of k and ε respectively

S_k is the user-defined source terms for k ($kg/(ms^2)$)

S_ε is the user-defined source terms for ε ($kg/(ms)$)

Table 4.1 shows the list of model constants and their values used in ANSYS FLUENT

The term for turbulence kinetic energy generation G_k is given by Equation (4.12).

The term for turbulence kinetic energy generation by buoyancy is given by

$$G_b = \beta_2 g_i \frac{\mu_t}{Pr_t} \frac{\partial T_e}{\partial x_i} \quad (4.40)$$

where,

Pr_t is turbulent Prandtl number for energy (for standard k - ε model value is 0.85)

g_i is gravitational vector's component in i^{th} direction (m/s^2)

β_2 is thermal expansion coefficient ($1/K$) and is given by

$$\beta_2 = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T_e} \right)_p \quad (4.41)$$

μ_t is the turbulent viscosity (Ns/m^2) and is given by

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (4.42)$$

$\frac{\partial T_e}{\partial x_i}$ is the temperature gradient

Dilation diffusion term is given by

$$Y_M = 2\rho \varepsilon M_t^2 \quad (4.43)$$

M_t is turbulent Mach number and is given by

$$M_t = \sqrt{\left(\frac{k}{a_1^2} \right)} \quad (4.44)$$

a_1 is the speed of the sound in (m/s)

$C_{1\varepsilon}$	$C_{2\varepsilon}$	C_μ	σ_κ	σ_ε
1.44	1.92	0.09	1.0	1.3

Table 4.1: Model constants for standard k- ε model

More detailed explanation of the transport equations can be found in the reference [5].

4.3.3. Large Eddy Simulation (LES)

4.3.3.1. Introduction

Turbulent flow is characterized by a wide range of length scale and time scale. In LES turbulence is divided into two parts

- Large Scale Turbulence (Large eddies)
- Small Scale Turbulence (Small eddies)

Large eddies are fully resolved, but the small eddies are modelled using the so called Sub Grid Scale (SGS) turbulence model [1].

4.3.3.2. Governing Equations for LES

The governing equations are obtained by the filtering the Equations (4.1) and (4.2). Thus, the filtering process filters the eddies which are smaller than the filter width size mentioned in the computations.

The general filtered variable (denoted by hat) is given by

$$\hat{\phi}_1(x) = \int_{D_1} \phi_1(x') G(x, x') dx' \quad (4.45)$$

where,

ϕ_1 is the general fluid variable (velocity or pressure)

D_1 is the fluid domain

$G(x, x')$ is the filter function that finds the resolved eddies scale

The default filtering operation used in the FLUENT finite volume discretization is

$$\hat{\phi}_1(x) = \frac{1}{V} \int_V \phi_1(x') dx', \quad x' \in V \quad (4.46)$$

Here, V is the volume of the cell (m^3)

Filter function G is given by

$$G(x, x') = \begin{cases} 1/V, & x' \in V \\ 0, & \text{otherwise} \end{cases} \quad (4.47)$$

The filtered governing equations can be derived from applying the filtering function to the Equations (4.1) and (4.2) as follows

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho \hat{u}_i) = 0 \quad (4.48)$$

$$\frac{\partial}{\partial t} (\rho \hat{u}_i) + \frac{\partial}{\partial x_j} (\rho \hat{u}_i \hat{u}_j) = \frac{\partial}{\partial x_j} \left(\mu \frac{\partial \hat{u}_i}{\partial x_j} \right) - \frac{\partial \hat{p}}{\partial x_i} - \frac{\partial \tau_{ij}^s}{\partial x_j} \quad (4.49)$$

τ_{ij}^s is the subgrid-scale stress (N/m^2) and is given by

$$\tau_{ij}^s = \rho \overbrace{u_i u_j} - \rho \hat{u}_i \hat{u}_j \quad (4.50)$$

where,

$\hat{u}$ is the filtered velocity (m/s)

$\hat{p}$ is the filtered pressure fields (Pa)

ρ is the density (kg/m^3)

μ is the dynamic viscosity ($N.s/m^2$)

4.3.3.2.1. Subgrid-scale models

The subgrid-scale stresses in the filtered Navier-Stokes equations are unknown and need to be modelled.

This term τ_{ij}^s is split into two parts as isotropic and deviatoric parts as given in Equation (4.51). The deviatoric part of the subgrid-scale stress tensor is modelled by widely used Smagorinsky model and is given by

$$\tau_{ij}^s = \underbrace{\tau_{ij}^s - \frac{1}{3}\tau_{kk}^s\delta_{ij}}_{\text{deviatoric}} + \underbrace{\frac{1}{3}\tau_{kk}^s\delta_{ij}}_{\text{isotropic}} \quad (4.51)$$

$$\tau_{ij}^s - \frac{1}{3}\tau_{kk}^s\delta_{ij} = -2\mu_{\text{SGS}}\hat{S}_{ij} \quad (4.52)$$

where,

μ_{SGS} is the subgrid-scale turbulent viscosity.

τ_{kk}^s is the isotropic part of the subgrid-scale stresses and is not modelled, but it can be added to the filtered static pressure term.

δ_{ij} is the Kronecker delta

$\hat{S}_{ij}$ is the rate of strain tensor for the resolved scale which is given by

$$\hat{S}_{ij} = \frac{1}{2} \left(\frac{\partial \hat{u}_i}{\partial x_j} + \frac{\partial \hat{u}_j}{\partial x_i} \right) \quad (4.53)$$

ANSYS Fluent offers four models for μ_{SGS} :

- Smagorinsky-Lilly model [1]
- Dynamic Smagorinsky-Lilly model [1]
- WALE model [1]
- Dynamic kinetic energy subgrid-scale model [1]

Smagorinsky-Lilly model is the most widely used model and is described as follows.

The eddy viscosity is defined by

$$\mu_{\text{SGS}} = \rho L_s^2 |\hat{S}| \quad (4.54)$$

where,

L_s is the mixing length for subgrid scales (m) and is given by

$$L_s = \min(K d, C_{s1} \Delta) \quad (4.55)$$

where,

K is the von Karman constant

d is the distance to the closest wall (m)

Δ is the local grid scale (m) and is computed by

$$\Delta = V^{1/3} \quad (4.56)$$

where,

V is the volume of the computational cell (m³)

C_{s1} is the Smagorinsky constant

$$|\hat{S}| = \sqrt{2\hat{S}_{ij}\hat{S}_{ij}} \quad (4.57)$$

C_{s1} is not a universal constant, which is the most important drawback of this model. Nevertheless, C_{s1} value of around 0.1 is set as default in ANSYS FLUENT because this value provides the best and suitable results for various range of flows [5]. The details of this and other models are given in the reference [5].

4.3.3.3. Introduction to Hybrid RANS/LES

RANS and LES both can be used to solve the governing equations (Equations (4.1) and (4.2)) by their respective ways. Table 4.2 compares the pros and cons of both RANS and LES. CFD has become the most powerful tool in the analysis of wakes in the wind turbine. RANS is commonly used (though still costly) to investigate the flow field in the wind farm. The RANS equation model solves with the mean wind velocities and turbulence kinetic energy being modelled throughout the domain. RANS is sufficient, for analyzing the general flow pattern and mean flow properties.

On the other hand, Large Eddy Simulation (LES) can be used to study in more detail about the flow-field in the wind farm. But the main barrier of using LES method is that it demands high computational cost and run-time memory.

RANS	LES
Less Accurate	More Accurate, still impractical for many flows due to intensive numerical Solution.
All the turbulent length scales are modelled	Only the large turbulent length scales are resolved. The smaller scales are modelled
Averaged version of the governing equations is solved along with another equation to represent all the turbulent scales	A filtered version of the governing equations is solved along with another equation to represent the turbulent small scales
Less expensive and time consuming compared to LES	More expensive and time consuming

Table 4.2: RANS Vs LES

Detached Eddy Simulation (DES) is the hybrid turbulence modelling that combines RANS and LES models. DES method can be considered as three-dimensional unsteady numerical simulation. In DES turbulence modelling RANS Model is used to analyze the near wall region while LES is used in the wake region where unsteady and chaotic flow is usually found. Thus, the computational time can be saved and compromised when compared to the usage of LES in the entire wind turbine. In the wake region, DES computes eddies and vortices according to LES analysis to maintain the dynamic features of the flow. This method also aims to better understand the complexity of separated flows

4.4. Modelling Approaches of Rotating Machineries using FLUENT

ANSYS FLUENT uses moving reference frame for problems involving moving parts such as rotating blades, impellers etc. In many situations, moving parts make the problem unsteady when it is viewed from stationary frame. But in moving reference frame, the flow around the moving part can be modelled as steady state with respect to the moving reference frame.

4.4.1. Single Reference Frame (SRF)

In SRF, entire computer domain is set to be moving reference frame. Single fluid domain rotates with constant speed about a particular axis. Walls which move with the domain

might be of any shape. Walls can also be defined as stationary but these walls must be surfaces of revolution about the axis of rotation. Figure 4.1 shows an example of SRF in which a rotor is rotating in a stationary wall. Rotationally periodic boundary conditions can also be used on the condition the surfaces must be periodic about the rotational axis. For more complex geometries, SRF approach may not be possible to use it. In such cases, the domains are divided into subdomains as explained in the next section [6].

Figure 4.1: An example of SRF model [7]

4.4.2. Multiple Reference Frame (MRF)

Multiple Reference Frame and Sliding Mesh approaches were applied to model the rotation of turbine rotor in ANSYS FLUENT. The fluid flow is analyzed either using rotating frame or stationary frame. MRF is applied to calculate time-averaged velocity and other turbulence quantities. In MRF analysis, the model is divided into few subdomains (at least two domains). Each domain has its own reference frame (stationary or rotating) and may rotate at different speeds. Typical configuration has two subdomains, rotor subdomain and stator subdomain. The rotor sub-domain has the rotating reference frame while the stator sub-domain has stationary reference frame. Despite this technique being approximation, it provides reasonably accurate results particularly when the interaction between the domains is low. The domain is divided in such a way that all the rotating walls are kept in the rotating sub-domain while the stationary walls in stationary sub-domain. The boundary between the two sub-domains is set to be interior zone or interface common to both, thus

that continuity is maintained. At the interfaces between cell zones, a local reference frame transformation is performed to enable flow variables in one zone to be used to calculate fluxes at the boundary of the adjacent zone [6]. Figure 4.2 illustrates the example of MRF analysis using a mixing tank with single impeller. The rotating reference frame is defined that encompasses the impeller and the fluid surrounding it and stationary frame is defined for the flow surrounding the impeller region. The dashes indicate the interface between the two frames. Mesh does not move in MRF analysis.

Figure 4.2: Mixing tank with single impeller [6]

4.4.3. Sliding Mesh (SM)

Sliding mesh is used when there is a time-dependent analysis, especially when solution is not getting properly converged. In this method rotating reference frame is physically rotating and this is done by rotating the rotor domain in small increments at each time step. The interface between the domains slides with one another as shown in Figure 4.3. Sliding mesh technique will give a better result compared to MRF technique [8].

Figure 4.3: Mesh motion in Sliding mesh approach [7]

CHAPTER 5: CFD SIMULATION OF WIND TURBINE

5.1. NREL Phase VI Turbine

5.1.1. Experimental data

Since the experimental data are needed for comparison and confirmation of the CFD Study, NREL Unsteady Aerodynamics Experiment (UAE) Phase VI Turbine analysis [1, 2] was selected. These tests were conducted in the year 2000 at the NASA Ames Research Centre wind tunnel (cross-sectional dimensions of the test section are $24.4 \times 36.6 \text{ m}^2$). It is the largest wind tunnel in the world. This wind tunnel has an ability to attain wind speeds up to 50 m/s. In this experiment, the wind speed range from 5 m/s (cut-in wind speed) to 25 m/s (cut-off wind speed) is used and turbulence intensity is maintained to be less than 0.5 % at the inlet. Figure 5.1 shows the rotor used in this experiment [1]. The rotor is a two-bladed stall regulated and the blades are tapered and twisted. The blades have S809 aerofoil section throughout the blade span from the root to the tip. The rotor is of 10.06m diameter. The nacelle is supported by a 0.4 m diameter tower at a hub height of 12m. The rotational speed of the rotor was fixed at around 72 RPM. Table 5.1 summarises experimental arrangements for different phases in the wind tunnel [1].

The rotor featured a rated power of 19.8 kW. Experiments were conducted in various configurations that include upwind and downwind operations, flat and coned, yawed and non-yawed, teetered and rigid hub, rotating and parked rotor. The description of rotor used for the CFD simulation in this study is summarised in Table 5.2 [1]. More details of the blade and the aerofoil S809 geometry are given in the Appendix E.

Cases	Air Density (kg/m ³)	Wind Speed (m/s)	Rotational Speed (RPM)
1	1.244	5.0	71.7
2	1.246	7.0	71.9
3	1.246	10.1	72.1
4	1.227	13.1	72.1
5	1.224	15.1	72.1
6	1.221	20.1	72.0
7	1.220	25.1	72.1

Table 5.1: Different cases of the NREL Phase VI experiment [1]

Figure 5.1: NREL Phase VI 2-bladed turbine in the NASA Ames Wind Tunnel [1]

Sequence	S
Number of blades	2
Rotor diameter	10.06 m
Rotor location	Upwind
Cone angle	Zero degree
Blade pitch angle	Three degree
Yawed angle	Zero degree
Blade sectional aerofoil profile	S809
Twist angle	Nonlinear twist along the blade span
Blade chord	0.737 (at $r = 1.257\text{m}$) to 0.358 (tip), linearly tapered
Blade thickness	20.95 % of chord (m)

Table 5.2: Rotor description of NREL Phase VI Turbine [1]

Since the blades are geometrically identical, in the experiment one of the blades is equipped with 22 pressure taps at five radial locations 30 %, 47%, 63%, 80%, 95% of the blade span as shown in Figure 5.2 (a) to have pressure measurements and subsequently calculate the aerodynamic force coefficients [1]. Also, pressure taps distribution around the aerofoil section at five radial stations has a larger concentration near the leading edge as shown in Figure 5.2 (b) [1]. In addition, two taps at the 4% and 36% chordwise direction were installed on the suction surface along the blade at different other intermediate locations as shown in Figure 5.2 (a) [1]. The blade is also equipped with 5-hole probes at 34%, 51%, 67%, 84%, and 91% of the blade span as shown in Figure 5.2 (a) [1].

(a)

(b)

Figure 5.2: Location of pressure taps on the blade [1]. (a) pressure tap distribution on the blade span (b) Pressure taps distribution along the aerofoil section at the prescribed radial locations

5.1.2. Aerodynamic Force Coefficients

Figure 2.27 of Chapter 2 shows aerodynamic force coefficients acting on the blade sections. The normal force coefficient (C_{NOR}) and tangential force coefficient (C_{TAN}) are calculated by integrating the pressure coefficient at certain sections.

The pressure coefficient is given by

$$C_p = \frac{p - p_\infty}{\left(\frac{1}{2}\right) \rho_\infty (U_\infty^2 + (\Omega r)^2)} \quad (5.1)$$

where,

p is the pressure (N/m²)

p_∞ is the freestream pressure (N/m²)

ρ_∞ is the freestream air density (kg/m³)

U_∞ is the wind speed (m/s)

Ω is the blade rotational speed (rad/s)

r is the radial distance from the hub centre to the blade section (m)

The normal and tangential force coefficients are given as

$$C_{NOR} = \sum_i \left(\frac{C_{p,i} + C_{p,i+1}}{2} \right) (x_{c,i+1} - x_{c,i}) \quad (5.2)$$

$$C_{TAN} = \sum_i \left(\frac{C_{p,i} + C_{p,i+1}}{2} \right) (y_{c,i+1} - y_{c,i}) \quad (5.3)$$

where,

x_c and y_c are normalized distance by the blade chord along the chord line (assumed to be x axis) and orthogonal to the chord line (assumed to be y axis) respectively. The value of x_c and y_c starts at the trailing edge ($x_c=1$), proceed along the upper surface and then the leading edge and proceed along the lower surface and ends in the starting point. While Equations (2.101) and (2.102) of Chapter 2 are in integral format, Equations (5.2) and (5.3) are in summation format as experimental data are extracted at discrete points.

5.2. CFD analysis of NREL Phase VI Turbine for the analysis of stall delay

Initially, CFD analysis was done using SST $k-\omega$ model in order to perform inverse BEM to compute $C_{L,3D}$ and $C_{D,3D}$ and improve the BEM analysis, which is discussed in Chapter 2.

5.2.1. Geometrical modelling

Based on the blade design information given in the Appendix E, three-dimensional geometric model of the blade was created using Solidworks. Initially MS Excel sheet is used to calculate the blade cross-sectional aerofoil data for a pitch angle 3° . Later curves of aerofoil were drawn in Solidworks and a feature called loft is used to create the solid part. Figures 5.3 (a and b) show aerofoil profiles at various sections around the twist axis and along the blade span respectively. Figure 5.4 shows the isometric view of the blade. Two blades were created and since the dimensions of the hub are not given, cylinder of diameter 0.218m (root hub adapter, see Appendix E) is created to connect the two blades. Several attempts were made with different domain sizes with both MRF and sliding mesh analysis. Finally, the analysis which provided promising results was done with sliding mesh analysis with rotor domain (thickness 3.5m) and stator domain of dimensions as shown in Figure 5.5 (a and b). Figure 5.5 (a) shows the front view and Figure 5.5 (b) shows the lateral view of both the domains. As discussed in Section 4.4.3 in Chapter 4, the boundaries at the contact of rotor domain and stator domain are indicated as interfaces through which information are exchanged.

(a)

(b)

Figure 5.3: Distribution of aerofoil profiles at various sections along the blade span. (a) distribution of aerofoils around the twist axis (b) distribution of aerofoils along the blade span

Figure 5.4: Isometric view of the blades

(a)

(b)

Figure 5.5: Computational domains for the CFD simulations. (a) front view (b) lateral view

5.2.2. Computational Mesh

Discretizing the domain into small shapes called meshing is the most important CFD application. It is difficult to create a fine mesh with an acceptable skewness value around the blades and to controlling the number of elements. The mesh can be structured, unstructured and hybrid. Structured mesh is homogeneous and contains the quadrilateral shape of the elements in two-dimensional mesh or the hexahedral shape of the elements in three-dimensional mesh, while unstructured mesh consists of the triangular shape of the elements in two-dimensional mesh or tetrahedral shape of the elements in three-

dimensional mesh. Hybrid mesh can either be a combination of structured mesh and unstructured mesh or combination of structured meshes of different element sizes.

The ANSYS meshing software was used to create an unstructured mesh. The blade is rounded to about 3mm at trailing edge in order to avoid error due to high skewness while running in ANSYS FLUENT. Inflation layer is created around the blade to accurately predict the flow around the boundary layer zone. The number of inflation layer was chosen to be 20 with a growth rate of 1.2. Since the Reynolds number is of range 1×10^6 and in order to obtain y^+ of around 1 to 2.5, the thickness of the first cell to the wall is kept at 2×10^{-5} m, based on the Equation (5.4) [3]. Unstructured mesh was generated with number of elements of approximately 43.6 million (10 million elements in the rotor domain and 33.6 million elements in the stator domain). Table 5.3 shows the maximum y^+ value computed on the blades using unsteady RANS (URANS) analysis for the wind speeds of 7, 10, 15, 20, 25m/s by using the sliding mesh technique. Detailed CFD results and their discussions are provided in Chapter 6. From the tabulated values, it is evident that the desired y^+ was obtained on choosing the inflation layer details described above.

$$\Delta y = Le \cdot y^+ \cdot \sqrt{74} \cdot Re^{\left(\frac{-13}{14}\right)} \quad (5.4)$$

where,

Δy is the distance of the first node from the wall (m)

Le is the characteristic length (m)

Re is the Reynolds number based on Le

y^+ is the dimensionless distance to the wall,

$$y^+ = \frac{y_1 u_*}{\nu} \quad (5.5)$$

y_1 is the coordinate normal to the wall or no slip boundary (m)

u_* is the friction velocity at the nearest wall (m/s)

$$u_* = \sqrt{\left(\frac{\tau_w}{\rho} \right)} \quad (5.6)$$

τ_w is the wall shear stress (N/m²)

ν is kinematic viscosity (m²/s)

ρ is density (kg/m³)

Cases	Inlet wind speed (m/s)	Maximum y^+ value
1	7.0	1.304
2	10	1.511
3	15	1.629
4	20	1.572
5	25	1.672

Table 5.3: Maximum y^+ values on blade surface for the selected wind speeds

5.2.3. FLUENT Solver set up and Simulation

The software ANSYS FLUENT solver is used to solve the governing Navier-Stokes equations with the aid of Finite Volume Method. Since it is an upwind turbine, the effects of the nacelle and tower are negligible and are excluded from this analysis. It is necessary to analyze the boundary conditions before setting up the FLUENT Solver.

5.2.3.1. Boundary Conditions

5.2.3.1.1. Velocity Inlet

The velocity at the inlet can be applied as either constant or variable (using UDF which will be explained in forthcoming section) and acting normal to the inlet or at a specified angle (as in the case of yawed flow). Since there is no yaw condition in our analysis, the uniform velocity is set normal to the inlet in the analysis. Inlet turbulence intensity is set to be maintained less than 0.5% and viscosity ratio (ratio between the turbulent viscosity to laminar viscosity) as 10 based on the default turbulence parameters of the NASA Ames

Wind Tunnel [4]. Hence in this CFD analysis, turbulence intensity used was 0.1% and viscosity ratio was 10.

5.2.3.1.2. Pressure Outlet

The gauge pressure of value zero (equal to the atmospheric pressure of 101,325 Pa) was applied at the outlet.

5.2.3.1.3. No slip wall

No slip wall boundary condition is applied to the blades. The blades are set to be moving walls. In this analysis, blade has to rotate at 72 RPM (7.53 rad/s) in anticlockwise direction.

5.2.3.1.3. Symmetry

The far-field is set to be Symmetry (zero shear slip walls).

Unsteady simulation was done in CPU with 4 cores and 64 GB RAM and processor speed of 3.70 GHz. Initially, the time step size was taken to be 0.1 seconds for the flow to reach from the inlet to the outlet. Then in order to get more accurate results, time step size was taken as 0.002315 seconds (time taken for blade to rotate 1 degree) and number of time steps was kept at 720 (two revolutions). It took almost 6 weeks (CPU running time) for each of the wind speed analysis.

5.3. CFD simulations of NREL Phase VI Turbine in different terrains

For these simulations k- ϵ turbulence model was used because this model was used to describe the ABL (discussed in Chapter 3).

5.3.1. Geometrical Modelling

For this analysis, no change was made on the blade and inner domain of the previous description. In order to be more realistic, outer domain was modelled with square cross section of size (50m x 50m) as shown in Figure 5.6. This would aid in applying inlet velocity varying with altitude.

Figure 5.6: 3D view of the domains set up for ABL analysis with dimensions

In addition, a nacelle and tower like structure were created in the outer domain behind the blades and hub height was kept to be 12m. Other dimensions are shown in Figure 5.6.

5.3.2. Pre-processing of ABL simulation analysis

ANSYS meshing software was used to mesh inner (rotor) domain. Inflation layer was created around the blade with first layer thickness 1×10^{-5} m. Maximum skewness was around 0.85 in the inner domain.

ANSYS ICEM software was used to mesh outer domain. The reason is explained later in this section. A mesh independent study was done for outer domain mesh in order to reduce the memory consumed and time taken while running in High Performance Computing (HPC) and also to study the boundary conditions of the top wall and also on the first layer thickness near to the ground. One of the important requirements of the ABL simulation is that distance of the centre point of the adjacent cell to the ground should not be less than the roughness height [5-7]. Equation (3.4) in Chapter 3 is reproduced here

$$k_s = \frac{9.793 * z_o}{C_s} \quad (5.7)$$

In FLUENT default value of roughness constant C_s is 0.5.

In this study, roughness length (z_o) used were 0.001m, 0.01m, 0.1m and 0.7m. As per Equation (5.7) on choosing $C_s=0.5$ (default value), the corresponding k_s values were 0.019586 m, 0.19586 m, 1.9586 m and 13.7102 m respectively. With the requirement or condition on the height of the centre point of the wall adjacent cells to the ground [5-7], the height of the wall adjacent cell should be at least 27.4204m (2×13.7102 m) for $z_o=0.7$ m which is almost half of the domain height (50 m). In general, this result in very coarse mesh for other cases also and in this specific study, it should be noted that hub height itself was 12m.

Alternately, the roughness constant (C_s) values can be increased to maintain the appropriate value of k_s to satisfy the requirements. In ANSYS FLUENT, C_s can be varied between 0 to 1 in Graphical User Interface (GUI). Hence, User Defined Function (UDF) was interpreted to set the corresponding roughness constant value.

One more important issue is the boundary condition of the top of the outer domain which will be discussed in the details of boundary conditions. Also the flow is highly turbulent, hence, it is necessary to perform the mesh study. But performing the study with turbine with MRF or sliding mesh will be time consuming and might lead to errors. Therefore, the simulations were done by making interfaces of the outer domain be wall (like disk-shaped building attached to the tower) and inner domain mesh is excluded. This is analogous to the CFD analysis of buildings in neutral ABL. A considerable number of simulations were done to choose the appropriate mesh for the outer domain. Table 5.4 shows the roughness length of the terrains chosen for the simulations, roughness height assumed and the height of the adjacent cell of the ground surface. Initially, attempts were made to create meshing using ANSYS default meshing software for FLUENT. This resulted in the modification of first adjacent of the ground near to the tower. Though the number of elements is low just around the tower are modified, it resulted in error while running in FLUENT. Therefore, a better meshing tool ANSYS ICEM was used. It created a good inflation layer (prism) for the adjacent mesh layer of the ground and also created uniform distributed unstructured mesh

in the outer domain. Thus, the number of mesh elements was considerably reduced to around 3 to 4 million.

Roughness Length z_o (m)	Roughness Height k_s (m)	Height of the adjacent layer to the ground (m)
0.7	0.15	0.3
0.1	0.15	0.3
0.01	0.05	0.1
0.001	0.05	0.1

Table 5.4: Parameters for creating outer domain mesh of the adjacent layer of the ground

5.3.3. FLUENT setup for wind turbine simulation in ABL

As discussed in Chapter 3, establishment of ABL profile is very important in order to study the wake analysis of the wind turbine.

5.3.3.1. Boundary Conditions

5.3.3.1.1. Velocity Inlet

In this analysis, velocity is not uniform but increases with altitude and also velocity profile depends on the roughness length z_o of the terrain. Hub height was chosen to be reference height and the reference velocity at hub height was assumed to be 10m/s.

Equations (3.1), (3.2) and (3.3) from Chapter 3 are reproduced here for better understanding of the homogenous neutral developed ABL flow at the inlet.

$$\bar{u}(z) = \frac{u_*}{K} \ln \left(\frac{z + z_o}{z_o} \right) \quad (5.8)$$

$$k = \frac{u_*^2}{\sqrt{C_\mu}} \quad (5.9)$$

$$\varepsilon = \frac{u_*^3}{K(z + z_o)} \quad (5.10)$$

It is necessary to know the friction velocity (u_*) in order to model the logarithmic velocity profile and its turbulent properties. It can be determined from the chosen reference height

(12m) and reference velocity (10m/s) from Equation (5.11) which is based on Equation (5.8).

$$u_* = \frac{K \times u_{ref} (z_{ref})}{\ln\left(\frac{z_{ref} + z_o}{z_{ref}}\right)} \quad (5.11)$$

where,

z_{ref} is the reference height above the ground (m)

u_{ref} is the reference velocity above the ground (m/s)

K is von Karman constant (=0.4)

u_* is friction velocity (m/s)

z_o is the roughness length of the terrain (m)

Table 5.5 shows the computed friction velocity for the chosen roughness length provided the reference height is 12m and the reference velocity is 10m/s. Appendix D includes UDF for inlet boundary conditions and also the roughness length for ground condition.

Roughness length z_o (m)	Reference height z_{ref} (m)	Reference velocity u_{ref} (m/s)	Friction velocity u_* (m/s)
0.7	12	10	1.40766
0.1	12	10	0.83551
0.01	12	10	0.56417
0.001	12	10	0.42586

Table 5.5: Values of friction velocity for different terrains

5.3.3.1.2. Ground surface

Ground was assumed to be no slip wall with roughness height based on the selected roughness length z_o as shown in Table 5.4 and Equation (5.7) was used to compute roughness constant which is applied on the ground using UDF.

5.3.3.1.3. Lateral walls or side walls

Symmetry boundary is used for the side walls (zero shear slip wall).

5.3.3.1.4. Outflow

The outflow boundary condition was used at the outlets. Based on experience, this boundary condition was preferred over pressure outlet to avoid errors while running the simulations.

5.3.3.1.5. Top wall (or top inlet)

Based on the experience gained from the mesh study, usage of symmetry boundary conditions (zero shear) on the top boundary resulted in error in velocity profile on approaching the turbine at top layers which is generally assumed to be not disturbed by the turbine due to high altitude. It also strongly recommended that constant shear stress has to be applied at the top boundary of the domain in order to maintain the ABL applied at inlet [5-7].

Shear stress is given by

$$\tau = \rho u_*^2 \quad (5.12)$$

Friction velocity at that altitude ($z=50\text{m}$) should be constant. Then from Equations (5.9) and (5.10), turbulence kinetic energy and turbulent dissipation rate values at that altitude should be constant. One approach is to assume the top boundary to be slip wall and specifying shear on it [5]. In this work, a different methodology was applied; instead of assuming it to be a wall, it is considered to be an inlet provided the flow is not normal to the boundary but in the component direction along the primary inlet ABL flow, by this approach velocity at top layer which is not going to be affected by wind turbine because of higher altitude is also fixed along with turbulence kinetic energy and turbulent dissipation rate. Table 5.6 shows mean velocity, turbulence kinetic energy and turbulent dissipation rate at height $z=50\text{m}$ for different terrains. These values are computed using the Equations (5.8), (5.9) and (5.10) and the values of friction velocity are in Table 5.5.

Roughness length z_o (m)	Mean velocity $\bar{u}$ (m/s)	Turbulence kinetic energy k (m²/s²)	Turbulent dissipation rate ε (m²/s³)
0.7	15.0221	6.60502	0.13946
0.1	12.9809	2.32693	0.0291
0.01	12.0128	1.06095	0.0089
0.001	11.5194	0.60453	0.0038

Table 5.6: Values applied at top boundary as inlet for different terrains

5.3.3.2. FLUENT simulation and considerations

Journal file (turbineinABL.jou) was written for complete sliding mesh analysis instead of creating the case file using the conventional GUI interface method. This journal file is attached in Appendix F. Since the simulations were done in HPC with allocated 8 nodes and 8 processors per node, it is convenient to run the journal file directly, instead of creating a case file and then a journal file to run in HPC.

Also one of the major issues faced is the error while running the simulations due to high turbulence values (especially with turbulent viscosity ratio exceeding the maximum limit). If this error occurs in one cell, it starts to multiply to other cells in a rapid manner resulting in termination of the simulation. A possible way to overcome this issue based on experience is to use first order discretization schemes or reduce the Under Relaxation Factor (URF). Though the simulation converges with above mentioned suggestions, velocity profile of the ABL and the results are not accurate. It is necessary that at least during final stage of simulations should be done in the second order discretization schemes and default URF values be used.

Initially, the time step size was taken to be 0.833 seconds (time taken for one complete revolution) for the flow to reach from the inlet to the outlet and first order discretization and reduced URF was used. Then in order to get more accurate results, time step size was taken as 0.002315 seconds (time taken for blade to rotate 1 degree) and number of time steps was kept at 720 (two revolutions). Then URF is increased slowly to default values

and last mentioned time step size and number of time steps were used for each increment in URF. Then discretization scheme is changed to second order and URF was lowered and simulation was done with time step size 0.002315 seconds and the number of time steps 720. Then URF is increased and finally, simulations were done with time step size 0.002315 seconds and the number of time steps was chosen 720 for the default second order scheme (except for turbulent dissipation rate) and default under relation factor. While performing in HPC, it is difficult to change these values each time and create a new case file and journal file and submit the job. Sometimes queue time taken will also be long. Thus, the journal file as attached in Appendix F can be submitted once and also reduce possible error resulting in termination of the simulations or even if termination occurs, previous saved case and data files can be retrieved and modified and can be resubmitted. Each simulation took around 5 to 6 days (CPU running time) in HPC@NTU. The number of nodes used for the simulation was 8 and processor per node is 8.

CHAPTER 6: RESULTS AND DISCUSSION

6.1. CFD results of NREL Phase VI Turbine for uniform inflow

The results of the URANS analysis of NREL Phase VI Turbine for uniform velocity at the inlet are initially discussed in this section, as these results will aid in the analysis of stall delay and inverse BEM. As discussed in Section 2.10 of Chapter 2, chordwise pressure coefficient distributions at different radial locations are necessary for the computation of aerodynamic forces at that particular blade section. Since the pressure distributions at 30%, 47%, 63%, 80% and 95% of blade span were measured during experimental aerodynamic analysis of the NREL Phase VI Turbine, these values are used for validation. Therefore, if there are any errors in the computed pressure coefficient values compared to experimental values, it will be regarded that the other parameters are also miscomputed. Experimental results were obtained on request from Dr. Scott Schreck, NREL. Around 15625 samples were measured for most variables in NREL experiments and averaged values of each variables are generally used for comparison.

Figures 6.1, 6.2, 6.3, 6.4 and 6.5 show the comparison of predicted URANS-CFD sectional pressure coefficients with time-averaged experimental measured pressure coefficients with its maximum and minimum values range at different radial locations 0.3R, 0.47R, 0.63R, 0.80R and 0.95R for wind speed 7, 10, 15, 20 and 25m/s respectively. Equation 5.1 is reproduced in this section as Equation 6.1 for better understanding on the computation of pressure coefficients.

$$C_p = \frac{p - p_\infty}{\left(\frac{1}{2}\right) \rho_\infty (U_\infty^2 + (\Omega r)^2)} \quad (6.1)$$

where,

$p - p_\infty$ is the static pressure difference between the measured pressure on the blade surface and freestream pressure (N/m²)

r is the radial distance from the hub centre to the blade section (m)

ρ_∞ is the freestream air density (kg/m³)

U_{∞} is the wind speed (m/s) and

Ω is the angular velocity of the blade and is equal to 7.53 rad/s

In general, pressure p on the pressure side of the blade surface (lower) will be greater than p_{∞} and in suction side (upper), pressure p will be less than p_{∞} . In order to avoid misunderstanding of the suction side (upper surface) and pressure side (lower surface), negative C_p is plotted against normalized distance (x/c) at that particular radial location as in Figures 6.1, 6.2, 6.3, 6.4 and 6.5.

In the experimental results (red coloured in the plot) provided by NREL for 7m/s wind speed, flow is attached in the suction side and there are fewer variations among the measured pressure of 15625 samples of each variable. From 10 m/s when the stall occurs at suction side, there is quite good variations among the measured pressure of 15625 samples of each variable in the suction side. Overall, URANS-CFD computed sectional pressure distributions in this work are close to the experimental results. However, some discrepancies are found at 47% blade span for 10 m/s and at 30% and 95% blade span for 15 m/s. Similar discrepancies were also noticed in other works such as RANS analysis of Sørensen et al. [1] and DES analysis of Li et al. [2]. Since the values at these five radial sections are predicted with good coincidence to the experimental results, it is considered that sectional pressure coefficients at another 13 span locations (0.25R, 0.267R, 0.3275R, 0.39R, 0.45R, 0.51R, 0.57R, 0.633R, 0.691R, 0.75R, 0.81R, 0.87R and 0.93R) provided in the design of NREL Phase VI Turbine should also be predicted well. Since the experimental measurements were not done in these sections, C_p plot is not included in this chapter for the additional sections taken in consideration to study stall delay. Instead Figures 6.6, 6.7, 6.8, 6.9 and 6.10 show pressure contour along with limiting streamlines on the pressure side and suction side of one of the two blades for wind speed 7 m/s, 10 m/s, 15 m/s, 20m/s and 25m/s respectively. As discussed in Section 2.9.2 of Chapter 2, it can be noticed that at lower wind speeds flow is attached at suction side and at higher wind speeds from 10m/s when the stall occurs, separated flow is moving radially which was considered

predominantly due to centrifugal force and the cause for the stall delay as discussed in Section 2.9.2 of Chapter 2. It can be also noticed on the suction side that the static pressure drops radially. This pressure gradient also assists the separated flow to move radially outwards.

Figures 6.11, 6.12, 6.13, 6.14 and 6.15 show the streamlines along the aerofoil sections at 30%, 47%, 63%, 80% and 95% of the blade radius of the wind speeds considered. At any sections, as the freestream velocity gets higher, angle of attack increases. This eventually results in flow separation that starts at trailing edge at few sections for freestream velocity of 10 m/s. Flow separation propagates towards the leading edge as the freestream velocity increases. This separated flow due to the mechanisms of stall delay as discussed in Section 2.9.2 of Chapter 2 gets transported towards the tip.

Figure 6.1: Pressure distributions for 7 m/s

Figure 6.2: Pressure distributions for 10 m/s

Figure 6.3: Pressure distributions for 15 m/s

Figure 6.4: Pressure distributions for 20 m/s

Figure 6.5: Pressure distributions for 25 m/s

Figure 6.6: Static pressure contours along with limiting streamlines on the pressure side (left) and suction side (right) for inlet wind speed 7m/s

Figure 6.7: Static pressure contours along with limiting streamlines on the pressure side (left) and suction side (right) for inlet wind speed 10m/s

Figure 6.8: Static pressure contours along with limiting streamlines on the pressure side (left) and suction side (right) for inlet wind speed 15m/s

Figure 6.9: Static pressure contours along with limiting streamlines on the pressure side (left) and suction side (right) for inlet wind speed 20m/s

Figure 6.10: Static pressure contours along with limiting streamlines on the pressure side (left) and suction side (right) for inlet wind speed 25m/s

$r/R=30\%$

$r/R=47\%$

$r/R=63\%$

$r/R=80\%$

$r/R=95\%$

Figure 6.11: Streamlines around different blade sections for inlet wind speed 7m/s

$r/R=30\%$

$r/R=47\%$

$r/R=63\%$

$r/R=80\%$

$r/R=95\%$

Figure 6.12: Streamlines around different blade sections for inlet wind speed 10m/s

$r/R=30\%$

$r/R=47\%$

$r/R=63\%$

$r/R=80\%$

$r/R=95\%$

Figure 6.13: Streamlines around different blade sections for inlet wind speed 15m/s

Figure 6.14: Streamlines around different blade sections for inlet wind speed 20m/s

Figure 6.15: Streamlines around different blade sections for inlet wind speed 25m/s

6.2. Computation of $C_{L,3D}$, $C_{D,3D}$ and AoA along the blade span by using Inverse BEM

Inverse BEM method as described in Section 2.10 (Figure 2.26) of Chapter 2 was applied to extract the sectional $C_{L,3D}$, $C_{D,3D}$ and AoA at 18 span locations along the blade for the probe of the stall delay effect. Chordwise pressure coefficients computed along different span locations were employed to compute C_{THRUST} and C_{TORQUE} based on Equations 2.103 and 2.104. It is necessary to consider sign convention while computing the forces from the pressure distributions.

Since C_L , C_D and AoA along the blade span cannot be directly measured from experiments as discussed in Section 2.10 of Chapter 2, Sørensen et al. [1], Johansen and Sørensen [3], Tangler and Kocurek [4], Sant et al. [5] and Schepers and van Rooij [6] computed these values at five radial locations with different techniques and their results are used for the validation. Figures 6.16 and 6.17 show the comparison of the extracted values at radial locations 0.3R, 0.47R, 0.63R, 0.8R and 0.95R using the inverse BEM method with other researchers. It can be noticed that computed $C_{L,3D}$ and $C_{D,3D}$ from improved inverse BEM methodology are in good agreement with the predictions of other researchers.

From the predicted $C_{L,3D}$ and $C_{D,3D}$ at each span location, AoA were computed as explained in Figure 2.25 of Chapter 2. Figures 6.18 and 6.19 show the computed $C_{L,3D}$ and $C_{D,3D}$ respectively along the blade span at 18 sections for the considered wind speeds. Since the values at the five radial locations considered during experiments are in good agreement with other works, as mentioned earlier, predictions at other 13 radial locations should also be correct. Figure 6.20 shows computed angles of attack along the blade the span for wind speeds 7m/s, 10m/s, 15m/s, 20m/s and 25m/s.

Figure 6.16: Comparison of extracted sectional $C_{L,3D}$ from different works with inverse BEM at five radial locations

Figure 6.17: Comparison of extracted sectional $C_{D,3D}$ from different works with inverse BEM at five radial locations

Figure 6.18: Computed $C_{L,3D}$ at 18 radial locations along the blade span using the inverse BEM method at different inlet wind speeds

Figure 6.19: Computed $C_{D,3D}$ at 18 radial locations along the blade span using the inverse BEM method at different inlet wind speeds

Figure 6.20: Angle of attack computed along the blade span using inverse BEM for different inlet wind speeds

The stall angle of attack (15.23°) indicated in Figure 6.20 is the 2D stall angle of attack taken from DUT experiments for $Re=10^6$. It can be noticed from Figure 6.11 that at 7 m/s flow in the suction side is attached to the blade and from Figure 6.20 AoA along the blade span is less than the 2D stall angle at all sections along the blade span. From Figures 6.12 and 6.20, it can be observed that for 10 m/s inlet wind speed, stall occurs at the lower half of the blade while near to the tip region, flows are still attached to the blade. At higher inlet wind speeds from 15 m/s onwards, it can be incurred from Figure 6.20 that the angles of attack are higher than the 2D stall angle of attack. Also, as the wind speed increases, AoA increases at the corresponding sections.

6.3. Comparison of predicted $C_{L,3D}$ and $C_{D,3D}$ from inverse BEM with values computed along the blade span in BEM analysis without stall delay model

As discussed in Section 2.9.3 of Chapter 2, BEM analysis was performed for the NREL Phase VI Turbine with C_L and C_D values of the aerofoil S809 extrapolated based on different proposals. Their power computations and comparison with experiments were discussed in that section. In this section, C_L and C_D values computed along the blade span in the BEM

analysis on using different aerofoil characteristics extrapolation methods are compared with the corresponding values determined using the inverse BEM methodology for NREL Phase VI Turbine.

Figures 6.21 and 6.22 present the computed $C_{L,3D}$ and $C_{D,3D}$ respectively along the blade span at 18 radial locations for wind speeds range from 7m/s to 25 m/s and their comparison with the predicted sectional C_L and C_D for each element in BEM analysis using aerofoil characteristics extrapolated by Viterna and Corrigan [7] model. It can be noticed in BEM analyses, at the inboard sections, the predicted sectional C_L and C_D which is necessary for torque or power computations do not correlate well with $C_{L,3D}$ and $C_{D,3D}$ computed using inverse BEM. Consequently, these models do not account for stall delay or they under-predict the lift coefficients at the inboard sections.

Figures 6.23 and 6.24 present the computed $C_{L,3D}$ and $C_{D,3D}$ respectively along the blade span at 18 radial locations using the inverse BEM method for wind speeds range from 7m/s to 25 m/s and their comparison with the predicted sectional C_L and C_D for each element in BEM analysis using aerofoil characteristics extrapolated by Lanzafame and Messina [8] model. As discussed in Section 2.9.3.2 and also from Figure 2.19 of Chapter 2, it can be observed that by using C_L at 45° AoA and C_D at 90° AoA for Reynolds number 3×10^5 in the Lanzafame and Messina [8] extrapolation model, the BEM computed power has good agreement with experimental values at lower wind speed (7 m/s) and higher wind speeds (20 m/s and 25 m/s). However, the BEM is not accurate in computing power while using the values of C_L at 45° AoA and C_D at 90° AoA at other Reynolds numbers (shown in Table 2.1) for extrapolation of aerofoil characteristics. It can be noticed in BEM analyses, at the inboard sections, the predicted sectional C_L and C_D do not correlate well with $C_{L,3D}$ and $C_{D,3D}$ computed using inverse BEM. Hence, these models do not account for stall delay as they under-predict the lift coefficients at the inboard sections.

Figures 6.25 and 6.26 present the computed $C_{L,3D}$ and $C_{D,3D}$ respectively along the blade span at 18 radial locations for wind speeds range from 7m/s to 25 m/s and their comparison with predicted sectional C_L and C_D for each element in BEM analysis using aerofoil characteristics extrapolation done by Tangler and Kocurek [9]. It can be understood that the predicted sectional C_L and C_D of BEM analysis along the blade span do not correlate well with $C_{L,3D}$ and $C_{D,3D}$ computed using inverse BEM making these models inefficient to account for stall delay. In other words, they under-predict the lift coefficients at the inboard sections of the blade.

Figure 6.21: Comparison of computed C_L from the inverse BEM and BEM analysis using Viterna-Corrigan model [7]

Figure 6.22: Comparison of computed C_D from the inverse BEM and BEM analysis using

Viterna-Corrigan model [7]

Figure 6.23: Comparison of computed C_L from the inverse BEM and BEM analysis using Lanzafame and Messina model [8]

Figure 6.24: Comparison of computed C_D from the inverse BEM and BEM analysis using

Lanzafame and Messina model [8]

Figure 6.25: Comparison of computed C_L from the inverse BEM and BEM analysis using Tangler and Kocurek model [9]

Figure 6.26: Comparison of computed C_D from the inverse BEM and BEM analysis using Tangler and Kocurek model [9]

6.4. Comparison of predicted $C_{L,3D}$ and $C_{D,3D}$ from inverse BEM with values computed along the blade span in BEM analysis with stall delay model.

As discussed in the previous section, BEM analysis without existing stall delay models and with different S809 aerofoil coefficients extrapolation models exhibit almost similar pattern for C_L and C_D distributions along the blade span. Therefore, in this section, Viterna and Corrigan [7] model (AR=11) was applied to extrapolate the aerofoil characteristics and used in BEM analysis incorporated with stall delay models developed by Snel et al. [10], Du and Selig [11], Chaviaropolous and Hansen [12] and Lindenburg [13] as discussed in Section 2.9.4 of Chapter 2. Figures 6.27 and 6.28 show the comparison of computed C_L and C_D values distribution along the blade span respectively on applying different stall delay models in BEM analysis with the predicted $C_{L,3D}$ and $C_{D,3D}$ from the inverse BEM for all the wind speeds considered. From Figure 2.24 of Chapter 2, it can be noticed that the power computed by BEM analysis on using existing stall delay correction models resulted in over-prediction of power. The reason behind the over-prediction of power by BEM on using stall delay models in Section 2.9.4 of Chapter 2 is discussed here. From Figure 6.27, it can be noticed that these models predict the increase in lift coefficient due to stall delay well up to wind speed 10m/s, but at higher wind speeds these models predict higher values of C_L compared to the values computed using inverse BEM method. In Figure 2.24 of Chapter 2, it can be noticed that power predicted by BEM on using Lindenburg [13] model is close to the experiment when compared to other models. This is due to the lift coefficients computed by Lindenburg [13] model being close to the values computed using inverse BEM when compared to the lift coefficients predicted using other models. From Figure 6.28, it is evident that stall delay models in general under-predict the C_D values especially at inboard sections. In BEM analysis, at each element, C_{TORQUE} is calculated by the following equation

$$C_{TORQUE} = C_L \sin(\phi) - C_D \cos(\phi) \quad (6.2)$$

Hence the rise in C_L value and fall in C_D values will eventually increase C_{TORQUE} , which in turn increases the power computed.

Without stall delay models, BEM methods under-predict the C_L values at inboard sections and with existing stall delay models, BEM over-predicts the power. Thus, it is necessary to develop a new method to incorporate the computed sectional $C_{L,3D}$ and $C_{D,3D}$ values in the BEM methodology for proper distribution of C_L and C_D values along the blade span.

Figure 6.27: Comparison of computed C_L from the inverse BEM and BEM analysis using different stall delay models

Figure 6.28: Comparison of computed C_D from the inverse BEM and BEM analysis using different stall delay models

6.5. Improved BEM analysis for stall delay by implementing $C_{L,3D}$ and $C_{D,3D}$ computed from Inverse BEM analysis

As noted earlier in Section 2.11 of Chapter 2, from 25% to 100% the blade span was divided into 15 sections with each section of length 5% of the blade radius. For BEM analysis, hub radius is taken to be 0.508m (0.1 R) and cylindrical section of the blade at the bottom extend from 0.508m (0.1 R) to 0.883m (0.176 R). From 0.883m (0.176 R) to 1.257m (0.25 R), there is a transition region from circular cross section of the cylindrical rod to S809 aerofoil. The schematic diagram is shown in Figure 2.29 of Chapter 2. Since the values of C_L and C_D at the inboard sections would be higher than the values at the tip sections and contribute for the torque prediction, hence the transition region cannot be neglected. Three additional span locations at 0.2R, 0.21R and 0.22R in the transition region which have almost S809 aerofoil were included in the BEM analysis. C_L and C_D for section I (shown in Figure 2.29 of Chapter 2) was used for the span locations in the transition region. In the new proposed BEM code, the blade was divided into 100 elements. Figures 6.29 and 6.30 present comparisons of C_L and C_D respectively along the blade span using the proposed BEM model with the results of inverse BEM for different wind speeds considered. It is apparent that the spanwise C_L and C_D distributions based on the proposed BEM method have good agreement with the values computed from inverse BEM.

It can be concluded that this new proposed BEM accounts for stall delay. By this manner, full rotor CFD simulation of a wind turbine can be performed and the inverse BEM method can be used to compute $C_{L,3D}$ and $C_{D,3D}$ which will account for stall delay. Hence, BEM can be improved and used in coupled BEM-CFD methods for the analysis of wind turbines on a farm.

Figure 6.29: Comparison of sectional C_L along the blade span from proposed BEM model with the values of CFD computed using inverse BEM

Figure 6.30: Comparison of sectional C_D along the blade span from proposed BEM model with the values of CFD computed using inverse BEM

6.6. Thrust, Torque and Power comparisons of NREL Phase VI Turbine (sequence S)

Figure 6.31 depicts the comparison of the thrust values computed from the proposed BEM model, CFD analysis and experimental values with their maximum and minimum range. Computed thrust values from BEM and CFD analysis agree well with the experimental values.

Figure 6.31: Comparison of thrust calculated at different wind speeds by different methods

Experimental results provided two torque values. One is the estimated aerodynamic torque which is computed from the pressure measurements and another is the measured Low Speed Shaft Torque (LSST). At lower wind speeds the averaged estimated torque and averaged LSST match well but at higher values, there is considerable deviation. Figure 6.32 (a) shows the comparison of torques computed via the proposed BEM model, CFD analysis with the experimental estimated aerodynamic torque plotted with its maximum and minimum range. While in Figure 6.32 (b), experimental LSST along with its range was used

to compare with the torque values computed from the proposed BEM model and CFD analysis.

(a)

(b)

Figure 6.32: Comparison of torque values calculated at different wind speeds. (a) comparison of values from proposed BEM model and CFD analysis with experimental estimated aerodynamic torque (b) comparison of values from proposed BEM model and CFD analysis with experimental LSST

Sørensen et al. [1] had also shown similar discrepancies at higher wind speed under heavily stalled conditions and explained that CFD simulations under-predict the torque by 20% in these conditions. It can also be noticed that estimated aerodynamic torque from pressure measurements are close to the torque values from the proposed BEM model and CFD analysis. In BEM model and CFD analysis, torque is computed based on the aerodynamic forces along the blade span which in turn calculated from the sectional pressure measurements. In the future, some corrections should be developed to account for the gap between the torque computed using pressure measurements and measured torque at higher wind speeds under heavily stalled condition.

Figure 6.33 shows the comparison of computed power values via the proposed BEM analysis, CFD and experimental values with its maximum to minimum range. Power values provided in the experimental results for different wind speeds were calculated via their corresponding LSST. Since torque values from the proposed BEM model and CFD analysis didn't fall within the range of LSST at higher wind speed (25 m/s). Power comparison also exhibit similar behavior because power is defined by the product of torque and angular velocity. Since the computation of angular velocity depends only on RPM which is fixed as 72, power depends only on the torque value.

Figure 6.33: Comparison of predicted power from BEM and CFD analyses and measured power from experiment

It is also observed that even CFD analysis cannot predict the torque values close to the measured LSST at heavily stalled condition, so it is difficult for BEM to achieve accurate results at higher wind speeds unless some corrections are made for torque computation.

6.7. Analysis of wake characteristics of NREL Phase VI Turbine

6.7.1. Wake characteristics for different uniform inflow velocities

In this section, discussion focus on the far wake kinematic models and turbulence intensity models described in Sections 3.3.3 and 3.4 of Chapter 3. CFD analysis is based on the descriptions in Section 5.2 of Chapter 5. Far wake analysis is more important in the wind farm design as described in Chapter 3. Rotor aerodynamics of NREL Phase VI Turbine were elaborated in detail in the previous sections of this chapter. Uniform inflow velocity of 7m/s, 10m/s, 15m/s, 20m/s and 25m/s were considered for the study of wake characteristics in this section. In each case, the corresponding uniform inlet velocity is taken as the reference velocity. As described in Section 3.3.2 of Chapter 3, near wake is assumed to be at the end of $(2 \times D)$ distance downstream of the rotor. Far wake kinematic models

described in Section 3.3.3 of Chapter 3 were used to compute the velocity deficit in the wake from the end of the near wake region ($2 \times D$).

6.7.1.1. Wake velocity profiles comparisons

The wake velocity is normalized with the reference velocity (uniform inflow velocity). Figure 5.5 of Chapter 5 demonstrates the computational domains and boundary conditions applied in this CFD analysis. Origin of the coordinate system was fixed at the hub centre (0, 0, 0). The rotational axis of the rotor is along the positive y axis. The downstream distance behind the wind turbine which is along the positive y axis and other length measurements along the other coordinate axes are normalized with the value of rotor diameter (D). In Chapter 3, x axis was used to represent the distance from the origin along the downstream wake direction. Since in the CFD analysis, uniform inflow velocity was applied along the y axis, for better understanding and interpretation of the results, 'y' would be considered as the distance from the origin along the downstream wake direction in the forthcoming explanations.

Normalized wake velocities are compared at the following normalized downstream distances (y/D) = 2, 5, 10, 15, 20, 25 and 30 for different uniform wind speeds considered. The radial distance from the origin (hub centre) is normalized with the rotor radius instead of rotor diameter for the better understanding of the wake width. The shape of the wake velocity profile, in general was shown in Figure 3.8 of Chapter 3. It can be observed from Figure 3.8 of Chapter 3 that at the near wake region velocity profile is 'w' shaped or double dip, while in the far wake region, velocity profile is Gaussian shape and gradually develops to the ambient velocity with the increase in downstream distance. Since uniform velocity was considered, wake velocity profiles should be axisymmetric. Figures 6.34, 6.35, 6.36, 6.37 and 6.38 show the comparison of the normalized wake velocity profiles from the URANS-CFD analyses with the normalized wake velocity profiles developed at the normalized downstream distances considered based on Jensen model [14], Larsen model [15], Frandsen model [16] and Ishihara model [17] for the wind speeds 7m/s 10m/s, 15m/s, 20m/s and 25m/s respectively. The wake velocity profiles generated using Jensen [14] and

Frandsen [16] models are hat shaped. These two models provide only a single value of the wake velocity predominantly based on the wake diameter and the downstream distance. There are no radial variations in the wake velocity at any downstream distance. While Larsen [15] and Ishihara [17] models provide Gaussian shaped velocity profile which is also a function of radial length. These four models also depend on the rotor thrust coefficient as mentioned in Table 3.2 of Chapter 3.

Figure 6.34: Far wake velocity profiles comparisons for uniform inflow 7m/s

Figure 6.35: Far wake velocity profiles comparisons for uniform inflow 10m/s

Figure 6.36: Far wake velocity profiles comparisons for uniform inflow 15m/s

Figure 6.37: Far wake velocity profiles comparisons for uniform inflow 20m/s

Figure 6.38: Far wake velocity profiles comparisons for uniform inflow 25m/s

Figure 6.39: Contours of U/U_{ref} in horizontal sectional plane taken at hub height

From Figures 6.34 to 6.38, it can be noticed that though both Jensen model [14] and Frandsen model [16] develop hat shaped wake velocity profiles, but the profiles developed based on Frandsen model [16] falls almost at the middle of the velocity profiles predicted from the CFD analysis except at very high wind speed 25 m/s. Also from Figures 6.34 to 6.38, it can be understood that wake diameter (D_w) predicted by Frandsen model [16] on comparison with Jensen [14] model is close to the CFD predicted values. Hence, Frandsen model [16] is comparatively better than Jensen model [14]. On comparing the wake velocity profiles created using Larsen model [15] and Ishihara model [17] with the CFD predicted velocity profiles, at low and moderate wind speeds such as 7m/s and 10m/s (Figures 6.34

and 6.35 respectively), these models predict a higher velocity deficit at centre line ($r=0$) for a normalized distance up to $(y/D)=5$. With the increment in the downstream distance, velocity profiles predicted by Ishihara model [17] are close to the CFD results, while Larsen model [15] slightly overestimates the velocity deficit. At high inflow wind speeds, such as 15m/s and 20m/s (Figures 6.36 and 6.37 respectively), from the normalized distance $(y/D)=2$ onwards the prediction of the velocity profiles developed by Larsen [15] model is closer to the CFD predictions than those by Ishihara model [17]. At higher wind speed, such as 25 m/s (Figure 6.38), for the normalized distance (y/D) upto 10, Larsen model [15] predicts velocity profiles closer to CFD while Ishihara model [17] over-predicts the velocity deficit at the centre line ($r=0$). At the normalized distance $(y/D)=15$, Larsen model [15] under-predicts while Ishihara model [17] over-predicts the velocity deficit on comparing with the CFD results. At the normalized distances $(y/D)=20$ and $(y/D)=25$, both Larsen model [15] and Ishihara model [17] under-predicts the velocity deficit on comparing with CFD results but the velocity profiles by Ishihara model [17] are closer to CFD compared to Larsen model [15]. At $(y/D)=30$, velocity profile developed from Larsen model [15] is close to CFD predictions while Ishihara model [17] over-predicts the velocity deficit. Figure 6.39 shows the velocity contour (U/U_{ref}) from the CFD results on the horizontal plane at the hub height. It can be also noticed from Figures 6.34 to 6.39, normalized velocity (U/U_{ref}) is between 0.925 and 0.975 at the centre even when (y/D) is 30. It indicates the wake velocity is not completely recovered (100%) to the freestream velocity even at this downstream distance. It is due to the low turbulence intensity of the inflow applied in the CFD analysis based on the experimental value. Since in these CFD analyses, uniform inflow velocity was applied and the effect of tower, wind shear etc. were not considered, hence the wake velocity profiles are axisymmetric. While the effect of wind shear cannot be neglected for the wake analysis of wind turbine in the ABL and will be discussed in Section 6.7.2 of this chapter.

6.7.1.2. Turbulence intensity comparison

In this CFD analysis, the turbulence intensity at the inlet was set to be 0.1% as an inlet boundary condition. Equation 3.41 of Chapter 3 was applied to compute turbulence intensity from turbulence kinetic energy and mean velocity and is expressed in percentage format. Figures 6.40, 6.41, 6.42, 6.43 and 6.44 show the turbulence intensity profiles predicted by the CFD analyses at the downstream distances $(y/D) = 0.25, 0.5, 1.5, 2, 5, 10, 15, 20, 25$ and 30 for the wind speeds $7\text{m/s}, 10\text{m/s}, 15\text{m/s}, 20\text{m/s}$ and 25m/s respectively. These figures also include the turbulence intensity at the inlet $((y/D)=-10)$ for the wind speeds considered. Turbulence intensities in the near wake region $((y/D)<2)$ are also analyzed. In the near wake region, the shape of the turbulence intensity profile is unpredictable. In these figures, $r/R=-1$ and 1 represents the blade tip. In the plot for $(y/D)=0.25$ in all these figures, at the region near to $r/R=\pm 1$ (blade tip regions), turbulence intensity drastically increases from the ambient into wake region. This indicates that the wake would be at higher turbulence compared to the ambient. A shear layer would be formed as explained in Section 3.2.1 of Chapter 3. The turbulence mixing occurs in the shear layer and the high turbulent low velocity wake mixes with the low turbulence high velocity freestream air in the shear layer. It is evident from these figures that due to turbulent mixing, turbulence intensity in the wake gradually reduces with the increment in the downstream distance. It was considered that the near wake region end at around $(y/D)=2$. In the far wake region after $(y/D)=2$, the turbulence intensity profile appears almost like Gaussian shaped though in few plots a very small dip is noticed at the centre line from the hub. Since the values are expressed in percentage format, deviation in the profile appear to be higher but it is however very low.

Figure 6.40: Turbulence intensity profiles at inlet and downstream distances of rotor for uniform inflow 7m/s

Figure 6.41: Turbulence intensity profiles at inlet and downstream distances of rotor for uniform inflow 10 m/s

Figure 6.42: Turbulence intensity profiles at inlet and downstream distances of rotor for uniform inflow 15 m/s

Figure 6.43: Turbulence intensity profiles at inlet and downstream distances of rotor for uniform inflow 20 m/s

Figure 6.44: Turbulence intensity profiles at inlet and downstream distances of rotor for uniform inflow 25m/s

It is evident from Figures 6.40 to 6.44 that turbulence intensity is not uniform at a particular downstream distance and it varies with radius. The models described in Section 3.4 of Chapter 3 provide only one value of turbulence intensity in the wake. Different empirical

models discussed in Chapter 3 are developed by Quarton and Ainslie [18], Hassan [19] and Larsen et al. [20]. Figure 6.45 shows the comparison of radially averaged turbulence intensity within the wake diameter at the concerned downstream distances predicted from the CFD with values from different empirical models provided in Section 3.4 of Chapter 3. On comparison with other models, Hassan model [19] provides closure results to the present CFD results at low wind speeds. Nevertheless, these models under-predict turbulence intensity even on comparing with the radially averaged wake turbulence intensity in the downstream distance from the CFD analyses. Hence these models fail in the prediction of turbulence intensity in the wake accurately.

Figure 6.45: Comparison of averaged turbulence intensity with in wake diameter and different existing models

6.7.2. Wake characteristics under different ABL

6.7.2.1. Inlet velocity profile

Prior to the analyses of the wake characteristics, it will be better to discuss on the velocity profile of the undisturbed neutrally stable ABL at the inlet. Unlike the previous study of uniform inflow, in this analysis, there would be a wind shear due to which the velocity varies with altitude. Hence, the reference velocity (U_{ref}) is chosen as 10 m/s which is at hub height ($z=12\text{m}$). Figure 6.46 shows the inlet velocity profile under the different ABL (roughness length $z_o=0.001\text{m}$, 0.01m , 0.1m and 0.7m). As shown in Figure 3.2 of Chapter 3, it is also observed in Figure 6.46 that the wind shear increases as the roughness of the ground increases. It incurs that on the rough surface, the blades above the hub height experiences relatively a higher velocity than blades below the hub height.

Since in this analysis, instead of uniform velocity, wind shear based on the roughness length was applied as an inlet boundary condition, wake velocity would not be axisymmetric. Hence both wake velocity profiles in the downstream distances are considered in this discussion.

Figure 6.46: Inlet velocity profiles for different ABL

6.7.2.2. Rotor aerodynamics under different ABL

Since the objective of this study is to couple between BEM-RANS, forces computed from the analytical method on the blades will be applied in the CFD analysis as an alternative to the full rotor simulation. Hence, it is necessary to analyze the rotor aerodynamics under different ABL conditions to confirm that there are no changes in the forces computed. The experiment and CFD results are available only for uniform flow of 10 m/s. Figure 6.47 shows the C_p comparisons at aerofoil blade sections along the blade span from the NREL Phase VI Turbine CFD simulations under different ABL with the values from the uniform inflow CFD and experiment analysis.

From Figure 6.47, it is evident that the C_p plots are almost similar along the blade span, the values fall within range of the experiment results except slight deviations are noticed on the leading-edge side. As discussed in Section 5.1.2 of Chapter 5, the aerodynamic forces are computed from the pressure values. If the pressure values correspond then the forces computed will also be in good agreement. A concern raised while performing the inverse BEM analysis to compute $C_{L,3D}$ and $C_{D,3D}$ from CFD results is that it must be repeated for different ABL. But it can be concluded that aerodynamic forces computed from the inverse BEM for uniform inflow can be used in coupled BEM-RANS model under different ABL for wind farm analysis.

Figure 6.47: Pressure comparisons along the blade span at selected radial locations between experiment results with uniform inlet flow of 10 m/s and CFD with uniform inflow and ABL with different roughness lengths.

6.7.2.3. Lateral profiles of normalized streamwise (axial) velocity along the horizontal sectional plane (hub height) at selected downstream distances

Figure 6.48 shows the lateral profiles of normalized streamwise velocity at the horizontal sectional plane taken at hub height ($z=12\text{m}$) under different ABL conditions at the selected downstream distances ($(y/D) = 2, 5, 10, 15, 20, 25$ and 30) and compared with the results of CFD analysis of uniform inlet velocity (10 m/s), which is described in the previous sections. Lateral profiles of normalized streamwise velocity at any particular downstream distance is almost two-dimensional axisymmetric along the centre line except the velocity profile for the roughness length $z_o=0.7\text{m}$ at the normalized downstream distance $(y/D)=30$.

It is evident that as the roughness length of the ground increases the wake is recovered at a faster rate. This is due to the higher rate of turbulent mixing. As the roughness length increases, ambient turbulence intensity increases based on Equation 3.40 and its aid in the turbulent mixing resulting in the wake velocity recovery. Figure 6.49 shows the normalized streamwise velocity (U/U_{ref}) contour along the horizontal sectional plane under different ABL conditions. It can also be observed in Figure 6.49 that the velocity is recovered at faster rate with increase in roughness length of the ground.

Figure 6.48: Horizontal lateral profiles of normalized streamwise velocity comparisons at different downstream distances under ABL with different roughness lengths and uniform inflow

Figure 6.49: Contours of U/U_{ref} along the horizontal plane under ABL with different roughness lengths

6.7.2.4. Vertical profiles of normalized streamwise (axial) velocity along the vertical midsection plane at selected downstream distances

Figure 6.50 shows the vertical profiles of normalized streamwise (axial) velocity at the midsection plane ($x=0$) along the vertical axis (z axis) under different ABL conditions at selected downstream distances ($(y/D) = 2, 5, 10, 15, 20, 25$ and 30) and compared with the results of uniform inlet velocity (10 m/s), which is described in the previous sections. Vertical profiles of normalized streamwise velocity are not axisymmetric due to change in the velocity at the inlet with altitude (wind shear). Figure 6.51 shows the contour of normalized

streamwise velocity (U/U_{ref}) at vertical midsection plane ($x=0$) under different ABL conditions. From Figures 6.50 and 6.51, it can be understood that wake velocity recovers at faster rate with the increase in ground roughness length.

Figure 6.50: Vertical profiles of normalized streamwise velocity comparisons along z axis at different downstream distances under ABL with different roughness lengths and uniform inflow

Figure 6.51: Contour of U/U_{ref} along the vertical midplane under ABL with different roughness lengths

6.7.2.5. Lateral profiles of turbulent intensity along the horizontal sectional plane (hub height) at inlet and selected downstream distances

Figures 6.52, 6.53, 6.54 and 6.55 show the lateral profiles of turbulence intensity along the horizontal plane at the hub height ($z=12m$) under different ABL conditions at the inlet ($y/D=-10$) and different selected downstream distances ($y/D=0.25, 0.5, 1.5, 2, 5, 10, 15, 20, 25$ and 30). From Figures 6.52 to 6.55, it can be noticed that at the inlet, the turbulence intensity increases with the roughness length of the ground. In the previous study under uniform flow and low ambient turbulence intensity, a peak is noticed at $r/R=1$ and -1 (tips

of the rotor). In this case, there is a sudden drop in the turbulence intensity value at the shear layer region in the near wake. In the far wake region, turbulence intensity is almost two-dimensional axisymmetric and centreline wake turbulence intensity value is lower than the ambient turbulence intensity. With the increase in the downstream distance, turbulence intensity gradually recovers towards the ambient condition.

Figure 6.52: Horizontal lateral profiles of turbulence intensity at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.001m

Figure 6.53: Horizontal lateral profiles of turbulence intensity at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.01m

Figure 6.54: Horizontal lateral profiles of turbulence intensity at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.1m

Figure 6.55: Horizontal lateral profiles of turbulence intensity at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.7m

6.7.2.6. Vertical profiles of turbulence intensity along the vertical midsection plane at inlet and selected downstream distances

Figures 6.56, 6.57, 6.58 and 6.59 show the vertical profiles of turbulence intensity along the vertical z axis at midplane (yz plane at $x=0$) for the different ABL conditions at the inlet ($(y/D) = -10$) and different downstream distances ($(y/D) = 0.25, 0.5, 1.5, 2, 5, 10, 15, 20, 25$ and 30). From Equation 3.41 of Chapter 3, it is evident that turbulence intensity increases with increase in turbulence kinetic energy and decreases with an increase in mean velocity. As explained in Section 6.7.1.2 of this chapter, in the absence of wind shear, turbulence intensity in the wake at downstream distance (especially at far wake) would be axisymmetric. In the analysis described in this section, mean wind speed varies with altitude and it influences the values of turbulence intensity.

In Figures 6.56, 6.57, 6.58 and 6.59 for all cases, wake turbulence intensity above the hub height recovers to the ambient inflow at faster rate compared to the wake turbulence intensity below the hub height. Chamorro and Porté-Agel [21] showed similar effects in their experimental analyses of the wake characteristics and explained the ground effect that this is ascribable to the presence of strong shear and helicoidal vortices in the upper half of the wake but in the lower part, the effect is due to the low mean shear and turbulence kinetic energy yield.

Peak value of turbulence intensity is at the hub height in all cases and it gradually reduces with the velocity recovery or turbulent mixing as proceeding downstream direction. It can be noticed that as the velocity deficit (shown in Figure 6.50) gradually recovers, turbulence intensity in the upper region also reduces and recovers to the ambient value. While in the lower region below hub height, turbulence intensity recovers by the addition in its value and rate at which it recovers depend on the terrain roughness characteristics. Hence it can be concluded that if a downstream turbine is placed at certain downstream distance where there is considerable difference in turbulence intensity above and below the hub height from the wake of the upstream turbine it can result in improper cyclic loading on blades of downstream turbine resulting in fatigue failure.

Figure 6.56: Vertical profiles of turbulence intensity along z axis at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.001m

Figure 6.57: Vertical profiles of turbulence intensity along z axis at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.01m

Figure 6.58: Vertical profiles of turbulence intensity along z axis at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.1m

Figure 6.59: Vertical profiles of turbulence intensity along z axis at inlet and downstream distances of the rotor and roughness length of ABL z_o is 0.7m

CHAPTER 7: CONCLUSION AND FUTURE WORKS

7.1. Conclusions

In this thesis, BEM theory is detailed and its limitations are described. One of the primary limitations of BEM is stall delay and the present study has attempted to improve the BEM analysis to account for stall delay. In general, BEM uses 2D aerofoil data, but in actual wind turbine because of the stall delay there will be an increase in aerofoil coefficients at inboard sections and it gradually reduces towards the tip. In order to study aerofoil coefficients, its methodology of polar extrapolation is necessary. The different methods for extrapolating the aerofoil characteristics of S809 are described. Different existing stall delay correction models in BEM analysis and their drawbacks are discussed. One possible solution is to compute 3D aerofoil characteristics from the direct rotor CFD analysis. Most researchers discussed in Section 2.10 of Chapter 2 have computed 3D aerofoil characteristics only at the five sections (0.3R, 0.47R, 0.63R, 0.80R, 0.95R) due to existing available experimental data. However, since CFD analysis does not have the same constraints as experiments, in this work 18 sections were considered for the computation of 3D aerofoil characteristics using improved BEM analysis. A new proposal for improving the BEM analysis to account for stall delay was discussed in Chapter 2. In the new proposal, aerofoil characteristics along the blade span in addition of being function of angle of attack and Reynolds number, vary radially. In this way, the BEM can account for stall delay.

In Chapter 3, wake aerodynamics of wind turbine under different ABL conditions were discussed. At first, atmospheric boundary layer and their properties were explained. Then, wake structure and properties of near wake and far wake regions were discussed. Finally, different existing models of wake velocity deficits and the turbulence intensity were detailed. The results of wake velocity models and wake turbulence intensity models were discussed in Chapter 6. An important conclusion of the wake study was that wake recovery depends on the turbulence in the atmospheric boundary layer. Streamwise turbulence intensity varies both vertically and laterally at all distances for the downstream rotor while existing

empirical models provide only one value of streamwise turbulence intensity at each downstream distance. In order to perform inverse BEM, initially rotor CFD analysis was done with uniform inflow velocity and SST $k-\omega$ turbulence model was used. Since the expressions for neutral ABL are based on $k-\varepsilon$ model, for wake aerodynamic analysis of the NREL Phase VI Turbine under different ABL, $k-\varepsilon$ model was used.

7.2. Future works for improving indirect rotor modelling (Coupling between BEM-RANS)

In wind farm analysis, computation of full rotor simulations of the array will be expensive and more time consuming. It is also not necessary to analyze the near wake region in farm analysis. The indirect rotor modelling is used for wind farm analysis in which forces induced by the rotor are directly applied to the momentum equation. There are different techniques which are briefly addressed in this section. In general, these models use 2D aerofoil data to compute the forces. Subsequent sections discuss the different models and possible ways of improving them based on the insights of this work.

(i) *Actuator disk model* is the simplest model. In this model, thrust forces computed from simple actuator disk method described in Section 2.2 or forces computed from BEM analysis described in Chapter 2 are used to compute pressure drop and applied to the disk. The blade is replaced with a thin disk. This model can be classified as AD-NR (Actuator disk non-rotating) and AD-R (actuator disk rotating). In ANSYS FLUENT, AD-NR model (no wake rotation) can be performed using 'porous jump boundary condition' by assuming the rotor disk to be porous or semipermeable disk and the pressure difference equation provided for porous jump boundary condition can be equated to the pressure drop equation computed via the induced forces from the momentum theory. This model is detailed in the references [1, 2].

From momentum theory described in Section 2.3 of Chapter 2, the relationship of pressure drop and velocity at rotor can be expressed as

$$p'' = \left(\frac{1}{2} \right) \rho \frac{U_{\text{rotor}}^2}{(1-a)} 4a \quad (7.1)$$

For porous jump, boundary condition pressure jump is given by

$$p'' = \left(\frac{\mu}{\alpha_{\text{porousmedia}}} U_{\text{rotor}} + \frac{C_2}{2} \rho U_{\text{rotor}}^2 \right) \Delta m \quad (7.2)$$

where,

p'' is pressure drop across the rotor or disk (Pa)

a is axial induction factor

U_{rotor} is air velocity at the rotor or disk (m/s)

Δm is the disk thickness (m)

ρ is density (kg/m³)

μ is dynamic viscosity (Ns/m²)

$\alpha_{\text{porousmedia}}$ is the permeability of porous media (m²)

C_2 is the pressure jump coefficient (1/m)

On comparing Equations (7.1) and (7.2) coefficients can be modelled and applied in porous media model in ANSYS FLUENT.

In ANSYS FLUENT, AD-R modelling can be formulated using 'fan boundary condition'. Again in 'fan boundary condition' the pressure difference equation provided for it can be equated with pressure difference Equation 7.1 to compute the required coefficients. This model, in addition, includes the implementation of tangential velocity component which accounts for wake rotation. This model is detailed in the reference [3].

Pressure jump in 'fan boundary condition' is provided by Equation 7.3

$$p'' = \sum_{n=1}^N f_n U_{\text{rotor}}^{N-1} \quad (7.3)$$

where,

f_n is polynomial coefficients

U_{rotor} is air velocity at the rotor or disk (m/s)

Polynomial coefficients (f_n) are modelled by correlating Equation (7.3) with Equation (7.1). From Equation (7.1) it is evident that Equation (7.3) should be a second order polynomial ($N=3$).

A possible way of improving these models is that instead of using disk, the disk can be divided into concentric rings with thickness equivalent to the section thickness in BEM analysis. Pressure drop (Equation (7.1)) can be computed from the elemental thrust or axial induction factor and U_{rotor} at that element and can be applied to the corresponding rings.

Another method of performing coupled BEM-RANS is adding the forces directly into the momentum equations in the disk by using the source term in ANSYS FLUENT [4, 5].

(ii) In *Actuator line models* [6], the blade is replaced by the line and the line is divided into elements. Induced forces (Equations (2.43) and (2.44)) computed from blade element theory described in Section 2.4 of Chapter 2 are applied to each element via adding the source term to the momentum equation at each element. This method is an improvement of AD model as the line represents the rotor. The main drawback of this model is flow around the aerofoil of full rotor cannot be produced by this method. In earlier works, C_L and C_D from 2D aerofoil data are used for the computation in Equations (2.39) and (2.40). The drawbacks of using 2D aerofoil data have been discussed in-depth in Chapter 2. Therefore, $C_{L,3D}$ and $C_{D,3D}$ computed from the inverse BEM method proposed in this work can be used to compute the induced forces.

(iii) Further improvement of the Actuator line model is *Actuator Surface (AS) model* [7, 8]. In this model blade is replaced by a surface along the chord and pressure difference is applied across the surface. Dobrev et al. [8] have used ‘fan boundary condition’ in ANSYS FLUENT to perform AS modelling of the NREL Phase VI Turbine. An iterative procedure is adopted in this model, in which BEM computes the pressure drop which is then applied to the CFD and the CFD flow field (velocities) results are returned to the BEM theory to compute pressure drop. It has been mentioned in this work that this model is not good for higher wind speeds. The possible reason might be that this model does not account for

stall (flow separation at suction side), the forces on the blade section cannot be computed correctly. An alternative way is to apply pressure difference directly computed from the improved proposed BEM method which accounts for the three-dimensional corrections in this model.

7.3. Future works for improving wind farm analysis

As discussed in Section 2.11 of Chapter 2 and Section 7.2, full rotor simulations of wind farm are not feasible, the indirect rotor model has to be used. It is evident from Chapter 3 and its corresponding results in Chapter 6, that ABL plays an important role in wake characteristics. Many past works describing indirect rotor models do not include the effect of different ABL conditions or comparisons are made with rotors fixed at certain directions and their wake velocity characteristics are compared with indirect rotor models.

Possible future work is to use the improved proposed BEM model in all these indirect rotor models described above under different ABL conditions and its results to be compared with direct rotor modelling as in Chapter 5. By this way, the best suited indirect models can be selected for wind farm analysis under different terrains.

7.4. Future works for improving turbulence intensity analysis in the wake

It was discussed that for ABL with higher roughness length, wake velocity is recovered at a shorter distance. However, that does not imply that the turbines can be placed close together because increase in turbulence intensity can result in structural failure of the turbine. It is also evident that turbulence intensity varies axially and radially. The turbulence intensity values also differ in the upper half and lower half of the hub height. It is also evident from the literature on indirect modelling that these models do not account for turbulence intensity computation. A possible solution is to perform many simulations for single turbine and analyze the wake characteristics as discussed in Chapters 5 and 6. Optimization techniques like Neural Networks and Genetic Programming can be used to model the

turbulence intensity. Again, these optimization models can be used for deriving the empirical relationships from the extensive results described previously.

REFERENCES

CHAPTER 1

- [1]. BP Group. (2011). BP Statistical Review of World Energy June 2011. London: BPstatus 2014 [2014-12-20].
- [2]. Annual Energy Outlook.(2010). Energy information administration. Department of Energy. United States
- [3]. USDoE. (2007). US department of energy on greenhouse gases. Department of Energy. United States
- [4]. Manwell, J. F., McGowan, J. G., & Rogers, A. L. (2009). Introduction: Modern Wind Energy and its Origins. Wind Energy Explained: Theory, Design and Application, 1-20.
- [5]. EWET. (2006). Wind Energy: A Vision for Europe in 2030. Advisory Council of the European Wind Energy Technology Platform.
- [6]. König, V. F. (1978). Windenergie in praktischer Nutzung. Verlag M. Pfriemer.
- [7]. Nelson, V. (2013). Wind energy: renewable energy and the environment. CRC press.
- [8]. Savonius, S. J. (1931). The S-rotor and its applications. Mech. Eng, 53, 333-338.
- [9]. Robert, E. W., Peter, B. S., & Stel, N. W. (1976). Aerodynamic performance of wind turbines. Oregon: Energy Research and Development Administration, Technical Information Center.

CHAPTER 2

- [1]. Robert, E. W., Peter, B. S., & Stel, N. W. (1976). Aerodynamic performance of wind turbines. Oregon: Energy Research and Development Administration, Technical Information Center.
- [2]. Vries, O. D. (1983). On the theory of the horizontal-axis wind turbine. Annual review of fluid mechanics, 15(1), 77-96.
- [3]. Hansen, A. C., & Butterfield, C. P. (1993). Aerodynamics of horizontal-axis wind turbines. Annual Review of Fluid Mechanics, 25(1), 115-149.
- [4]. Burton, T., Sharpe, D., Jenkins, N., & Bossanyi, E. (2001). Wind energy handbook. John Wiley & Sons.
- [5]. Hansen, M. O. L. (2008). Aerodynamics of Wind Turbines: second edition. (2 ed.) Earthscan Publications Ltd.
- [6]. Kulunk, E. (2011). Aerodynamics of wind turbines. INTECH Open Access Publisher.

- [7]. Wilson, R. E. (1980). Wind-turbine aerodynamics. *Journal of Wind Engineering and Industrial Aerodynamics*, 5(3-4), 357-372.
- [8]. Jonkman, J. M. (2003). Modeling of the UAE Wind Turbine for Refinement of FAST_AD (No. NREL/TP-500-34755). National Renewable Energy Lab., Golden, CO (US).
- [9]. Døssing, M., Madsen, H. A., & Bak, C. (2012). Aerodynamic optimization of wind turbine rotors using a blade element momentum method with corrections for wake rotation and expansion. *Wind Energy*, 15(4), 563-574.
- [10]. Lanzafame, R., & Messina, M. (2007). Fluid dynamics wind turbine design: Critical analysis, optimization and application of BEM theory. *Renewable energy*, 32(14), 2291-2305.
- [11]. Rajakumar, S., & Ravindran, D. (2012). Iterative approach for optimising coefficient of power, coefficient of lift and drag of wind turbine rotor. *Renewable energy*, 38(1), 83-93.
- [12]. Madsen, H. A., Mikkelsen, R., Øye, S., Bak, C., & Johansen, J. (2007). A Detailed investigation of the Blade Element Momentum (BEM) model based on analytical and numerical results and proposal for modifications of the BEM model. In *Journal of Physics: Conference Series* (Vol. 75, No. 1, p. 012016). IOP Publishing.
- [13]. Vaz, J. R. P., Pinho, J. T., & Mesquita, A. L. A. (2011). An extension of BEM method applied to horizontal-axis wind turbine design. *Renewable Energy*, 36(6), 1734-1740.
- [14]. Moriarty, P. J., & Hansen, A. C. (2005). AeroDyn Theory Manual (No. NREL/TP-500-36881). National Renewable Energy Laboratory., Golden, CO (US).
- [15]. Shen, W. Z., Mikkelsen, R., Sørensen, J. N., & Bak, C. (2005). Tip loss corrections for wind turbine computations. *Wind Energy*, 8(4), 457-475.
- [16]. Glauert, H. (1926). The analysis of experimental results in the windmill brake and vortex ring states of an airscrew. HM Stationery Office.
- [17]. Glauert, H. (1926). The elements of aerofoil and airscrew theory. Cambridge University Press.
- [18]. Spera, D. A. (1994). Wind turbine technology. ASME Press.
- [19]. Buhl, M. L. (2005). A new empirical relationship between thrust coefficient and induction factor for the turbulent windmill state (No. NREL/TP-500-36834) . Golden, CO: National Renewable Energy Laboratory.
- [20]. Leishman, J. G. (2002), Challenges in modelling the unsteady aerodynamics of wind turbines. *Wind Energy*, 5, 85–132.
- [21]. Hansen, A. C. (1992). Yaw dynamics of horizontal axis wind turbines (No. NREL/TP-442-4822). National Renewable Energy Lab., Golden, CO (US).

- [22]. Snel, H., & Schepers, J. G. (1995). Joint investigation of dynamic inflow effects and implementation of an engineering method. Netherlands Energy Research Foundation ECN.
- [23]. Schepers, J. G., & Snel, H. (1995). JOULE2: Dynamic Inflow: Yawed Conditions and Partial Span Pitch. Energy Research Center of the Netherlands, ECN-C-95-056.
- [24]. Sant, T. (2007). Improving BEM-based aerodynamic models in wind turbine design codes. TU Delft, Delft University of Technology.
- [25]. Schepers, J. G. (2012). Engineering models in wind energy aerodynamics: Development, implementation and analysis using dedicated aerodynamic measurements. TU Delft, Delft University of Technology.
- [26]. Leishman, J. G., & Beddoes, T. S. (1989). A Semi - Empirical Model for Dynamic Stall. *Journal of the American Helicopter society*, 34(3), 3-17.
- [27]. Øye, S. (1991, January). Dynamic stall simulated as time lag of separation. In *Proceedings of the 4th IEA Symposium on the aerodynamics of wind turbines*.
- [28]. Hansen, M. H., Gaunaa, M., & Madsen, H. A. A. (2004). A Beddoes-Leishman type dynamic stall model in state-space and indicial formulations, Report No. R-1354 (EN), Risø National Laboratory.
- [29]. Pereira, R., Schepers, G., & Pavel, M. D. (2013). Validation of the Beddoes–Leishman dynamic stall model for horizontal axis wind turbines using MEXICO data. *Wind Energy*, 16(2), 207-219.
- [30]. Liu, P., Yu, G., Zhu, X., & Du, Z. (2014). Unsteady aerodynamic prediction for dynamic stall of wind turbine airfoils with the reduced order modeling. *Renewable Energy*, 69, 402-409
- [31]. Guntur, S., Sørensen, N. N., Schreck, S., & Bergami, L. (2016). Modeling dynamic stall on wind turbine blades under rotationally augmented flow fields. *Wind Energy*, 19(3), 383-397.
- [32]. Himmelskamp, H. (1947) Profile Investigations on a Rotating Airscrew. MAP Volkenrode Report and Translation No. 832
- [33]. Banks, W. H. H., & Gadd, G. E. (1963). Delaying effect of rotation on laminar separation. *AIAA Journal*, 1(4), 941-942
- [34]. Dwyer, H. A., & McCroskey, W. J. (1971). Crossflow and unsteady boundary-layer effects on rotating blades. *AIAA Journal*, 9(8), 1498-1505.
- [35]. Milborrow, D. J. (1985). Changes in aerofoil characteristics due to radial flow on rotating blades. In *Proceedings of the 7-th BWEA conference*

- [36]. Madsen, H., & Christensen, H. F. (1990). On the relative importance of rotational, unsteady and three-dimensional effects on the HAWT rotor aerodynamics. In W. Palz (Ed.), European Community wind energy conference. Proceedings. (Vol. EUR-13251, pp. 227-232). Bedford: H.S. Stephens and Associates.
- [37]. Ronsten, G. (1992). Static pressure measurements on a rotating and a non-rotating 2.375 m wind turbine blade. Comparison with 2D calculations. *Journal of Wind Engineering and Industrial Aerodynamics*, 39(1), 105-118.
- [38]. Wood, D. H. (1991). A three-dimensional analysis of stall-delay on a horizontal-axis wind turbine. *Journal of Wind Engineering and Industrial Aerodynamics*, 37(1), 1-14.
- [39]. Narramore, J. C., & Vermeland, R. (1992). Navier-Stokes calculations of inboard stall delay due to rotation. *Journal of Aircraft*, 29(1), 73-78.
- [40]. Snel, H., Houwink, R., & Bosschers, J. (1994). Sectional prediction of lift coefficients on rotating wind turbine blades in stall. Petten, Netherlands: Netherlands Energy Research Foundation.
- [41]. Simms, D., Schreck, S., Hand, 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, Report No. NREL/TP-500-29494.
- [42]. Schreck, S., & Robinson, M. (2002). Rotational augmentation of horizontal axis wind turbine blade aerodynamic response. *Wind Energy*, 5, 133-150.
- [43]. Breton, S. P., Coton, F. N., & Moe, G. (2008). A study on rotational effects and different stall delay models using a prescribed wake vortex scheme and NREL phase VI experiment data. *Wind Energy*, 11(5), 459-482.
- [44]. Pereira, R. (2010). Validating the Beddoes–Leishman Dynamic Stall Model in the Horizontal Axis Wind Turbine Environment. (M.Sc dissertation, Thesis, Department of Control and Operations, Delft University of Technology, Delft, The Netherlands).
- [45]. Lindenburg, C. (2003). Investigation into rotor blade aerodynamics. Netherlands Society for Energy and the Environment, Paper ECN-C-03-025.
- [46]. Corten, G. P. (2001). Flow separation on wind turbines blades. Ph.D. Dissertation 2001; University of Utrecht : Rotterdam, Netherlands.
- [47]. Lindenburg, C. (2004). Modelling of rotational augmentation based on engineering considerations and measurements. In European Wind Energy Conference (EWEC 2004), London, UK, Nov (pp. 22-25).
- [48]. Hu, D., Hua, O., & Du, Z. (2006). A study on stall-delay for horizontal axis wind turbine. *Renewable Energy*, 31(6), 821-836.

- [49]. Breton, S. P. (2008). Study of the stall delay phenomenon and of wind turbine blade dynamics using numerical approaches and NREL's wind tunnel tests. (Doctoral theses at NTNU)
- [50]. Butterfield, C. P., Simms, D., Musial, W., & Scott, G. (1990). Spanwise aerodynamic loads on a rotating wind turbine blade (No. SERI/TP-257-3983; CONF-9009107--4). SRI International, Menlo Park, CA (USA).
- [51]. Butterfield, C. P., Huyer, S., & Simms, D. A. (1992). Recent results from data analysis of dynamic stall on wind turbine blades. National Renewable Energy Laboratory.
- [52]. Sørensen, N. N., Michelsen, J. A., & Schreck, S. (2002). Navier – Stokes predictions of the NREL phase VI rotor in the NASA Ames 80 ft × 120 ft wind tunnel. *Wind Energy*, 5(2 - 3), 151-169.
- [53]. Viterna, L. A., & Corrigan, R. D. (1982). Fixed pitch rotor performance of large horizontal axis wind turbines. DOE/NASA Workshop on Large Horizontal Axis Wind Turbines, Cleveland, OH, 1981; 69–85.
- [54]. Tangler, J., & Kocurek, J. D. (2005). Wind turbine post-stall airfoil performance characteristics guidelines for blade-element momentum methods. In 43rd AIAA Aerospace Sciences Meeting and Exhibit (pp. 10-13). Reno, Nevada.
- [55]. Du, Z., & Selig, M. (1998). A 3-D stall-delay model for horizontal axis wind turbine performance prediction. In 1998 ASME Wind Energy Symposium (p. 21).
- [56]. Chaviaropoulos, P., & Hansen, M. (2000). Investigating three-dimensional and rotational effects on wind turbine blades by means of a quasi-3D Navier-Stokes solver. *Journal of fluids engineering*, 122(2), 330-336.
- [57]. Johansen, J., & Sørensen, N. N. (2004). Aerofoil characteristics from 3D CFD rotor computations. *Wind Energy*, 7(4), 283-294.
- [58]. Sant, T., van Kuik, G., & van Bussel, G. J. W. (2006). Estimating the angle of attack from blade pressure measurements on the NREL phase VI rotor using a free wake vortex model: Axial conditions. *Wind Energy*, 9(6), 549-577.
- [59]. Schepers, J. G., & van Rooij, R. P. J. O. M. (2008). Analysis of aerodynamic measurements on a model wind turbine placed in the NASA-Ames tunnel. Energy Research Centre of the Netherlands ECN.
- [60]. Guntur, S., & Sørensen, N. N. (2014). An evaluation of several methods of determining the local angle of attack on wind turbine blades. In *Journal of Physics: Conference Series* (Vol. 555, No. 1, p. 012045). IOP Publishing.
- [61]. Hand, M. M., Simms, D. A., Fingersh, L. J., Jager, D. W., Cotrell, J. R., Schreck, S., & Larwood, S. M. (2001). Unsteady aerodynamics experiment phase VI: wind tunnel

test configurations and available data campaigns (No. NREL/TP-500-29955). National Renewable Energy Laboratory, Golden, CO(US).

CHAPTER 3

- [1]. Bhatia, S. C. (Ed.). (2014). Advanced Renewable Energy Systems,(Part 1 and 2). CRC Press.
- [2]. Barry, R. G., & Chorley, R. J. (2009). Atmosphere, weather and climate. Routledge.
- [3]. Huddart, D., & Stott, T. (2013). Earth environments: past, present and future. John Wiley & Sons.
- [4]. Blocken, B. (2014). CFD in building engineering: Fundamentals and applications in urban physics and wind engineering. Course book.
- [5]. Stull, R. B. (2012). An introduction to boundary layer meteorology (Vol. 13). Springer Science & Business Media
- [6]. Isyumov, N. (2012). Alan G. Davenport's mark on wind engineering. Journal of Wind Engineering and Industrial Aerodynamics, 104, 12-24
- [7]. Burton, T., Sharpe, D., Jenkins, N., & Bossanyi, E. (2001). Wind energy handbook. John Wiley & Sons.
- [8]. Doswell III, C. A. (1991). A review for forecasters on the application of hodographs to forecasting severe thunderstorms. Natl. Wea. Dig, 16(1), 2-16.
- [9]. Pena Diaz, A. (2009). Sensing the wind profile. Technical University of DenmarkDanmarks Tekniske Universitet, Risø National Laboratory for Sustainable EnergyRisø Nationallaboratoriet for Bæredygtig Energi.
- [10]. Sucevic, N., & Djuricic, Z. (2012). Influence of atmospheric stability variation on uncertainties of wind farm production estimation. In European Wind Energy Conference and Exhibition
- [11]. Richards, P. J., & Hoxey, R. P. (1993). Appropriate boundary conditions for computational wind engineering models using the k- ϵ turbulence model. Journal of wind engineering and industrial aerodynamics, 46, 145-153.
- [12]. Blocken, B., Stathopoulos, T., & Carmeliet, J. (2007). CFD simulation of the atmospheric boundary layer: wall function problems. Atmospheric environment, 41(2), 238-252.
- [13]. Hargreaves, D. M., & Wright, N. G. (2007). On the use of the k- ϵ model in commercial CFD software to model the neutral atmospheric boundary layer. Journal of Wind Engineering and Industrial Aerodynamics, 95(5), 355-369.
- [14]. Bertin, J. J., & Smith, M. L. (1998). Aerodynamics for engineers: 1 ed. Englewood Cliffs, Prentice Hall.

- [15]. Vermeer, L. J., Sørensen, J. N., & Crespo, A. (2003). Wind turbine wake aerodynamics. *Progress in aerospace sciences*, 39(6), 467-510.
- [16]. Gómez-Elvira, R., Crespo, A., Migoya, E., Manuel, F., & Hernández, J. (2005). Anisotropy of turbulence in wind turbine wakes. *Journal of Wind Engineering and Industrial Aerodynamics*, 93(10), 797-814.
- [17]. Ivanell, S., Mikkelsen, R., Sørensen, J. N., & Henningson, D. (2008). Three-dimensional actuator disc modeling of wind farm wake interaction. *interaction*, 1(2), 3.
- [18]. Hansen, M. O. L. (2008). *Aerodynamics of Wind Turbines: second edition*. (2 ed.) Earthscan Publications Ltd.
- [19]. Marmidis, G., Lazarou, S., & Pyrgioti, E. (2008). Optimal placement of wind turbines in a wind park using Monte Carlo simulation. *Renewable energy*, 33(7), 1455-1460.
- [20]. Johnson, K. E., & Thomas, N. (2009). Wind farm control: addressing the aerodynamic interaction among wind turbines. In *2009 American Control Conference* (pp. 2104-2109). IEEE.
- [21]. Sanderse, B. (2009). *Aerodynamics of wind turbine wakes: literature review*. ECN 2009. Report ECN-E-09e016.
- [22]. Barthelmie, R. J., Hansen, K., Frandsen, S. T., Rathmann, O., Schepers, J. G., Schlez, W., Phillips, J., Rados, K., Zervos, A., Politis, E.S., & Chaviaropoulos, P. K. (2009). Modelling and measuring flow and wind turbine wakes in large wind farms offshore. *Wind Energy*, 12(5), 431-444.
- [23]. Adaramola, M. S., & Krogstad, P. Å. (2011). Experimental investigation of wake effects on wind turbine performance. *Renewable Energy*, 36(8), 2078-2086.
- [24]. Krogstad, P. Å., & Lund, J. A. (2012). An experimental and numerical study of the performance of a model turbine. *Wind Energy*, 15(3), 443-457.
- [25]. Krogstad, P. Å., & Adaramola, M. S. (2012). Performance and near wake measurements of a model horizontal axis wind turbine. *Wind Energy*, 15(5), 743-756.
- [26]. Emeis, S. (2010). Meteorological explanation of wake clouds at Horns Rev wind farm. *Dewi Magazin*, 37, 52-55.
- [27]. Kay, A., & Lee, W.K. (2014). Characterisation of Downstream flow from Large Wind Turbines(No.2013/493). National Measurement office, Middlessex (UK).
- [28]. Ross, J. N., & Ainslie, J. F. (1981). Wake measurements in clusters of model wind turbines using laser Doppler anemometry. In *Proceedings of the Third BWEA Wind Energy Conference*, Cranfield (pp. 172-84).

- [29]. Vermeulen, P. E. J., & Builtjes, P. J. H. (1982). Turbulence Measurements in Simulated Wind-Turbine Clusters. Netherlands Organization for Applied Scientific Research, Division of Technology for Society..
- [30]. Hassan, U., Glendinning, A. G., & Morgan, C. A. (1990). A wind tunnel investigation of the wake structure and machine loads within small wind turbine farms. In Proceedings of the 12th BWEA Wind Energy Conference. Norwich, UK (pp. 47-52).
- [31]. Smith, D. (1990). Multiple wake measurements and analysis. In Proceedings of the 12th BWEA Wind Energy Conference. UK: Norwich (pp. 53-6).
- [32]. Smith, D., & Taylor, G. J. (1991). Further analysis of turbine wake development and interaction data. In Proceedings of the 13th BWEA Wind Energy Conference, Swansea, UK (pp. 325-331).
- [33]. Taylor, G. J., & Smith, D. (1991). Wake measurements over complex terrain. In Proceedings of the 13th BWEA Wind Energy Conference, Swansea, UK (pp. 335-42).
- [34]. Crespo, A., & Hernandez, J. (1993). Analytical correlations for turbulence characteristics in the wakes of wind turbines. In European Community Wind Energy Conference, Travemuende, Germany, March (pp. 8-12).
- [35]. Baker, R. W., & Walker, S. N. (1982). Wake studies at the Goodnoe Hills MOD-2 site (No. DOE/BP-182; BPA-82-11). Oregon State Univ., Corvallis (USA). Dept. of Atmospheric Science.
- [36]. Taylor, G. J., Milborrow, D. J., McIntosh, D. N., & Swift-Hook, D. T. (1985). Wake measurements on the Nibe windmills. In Proceedings of the 7th BWEA Wind Energy Conference, Oxford, UK (pp. 67-73).
- [37]. Höglström, U., Asimakopoulou, D. N., Kambezidis, H., Helmis, C. G., & Smedman, A. (1988). A field study of the wake behind a 2 MW wind turbine. *Atmospheric Environment* (1987), 22(4), 803-820.
- [38]. Frandsen, S. T., Chacón, L., Crespo, A., Enevoldsen, P., Gómez-Elvira, R., Hernández, Højstrup . J., Manuel, F., & Thomsen, K. (1996). Measurements on and modelling of offshore wind farms.
- [39]. Rados, K., Larsen. G., Barthelmie, R., Schlez, W., Lange, B., Schepers, G., Hegberg, T., & Magnisson, M. (2001). Comparison of wake models with data for offshore windfarms. *Wind Engineering*, 25(5), 271-280.
- [40]. Øye, S. (1990). A simple vortex model. In Proc. of the third IEA Symposium on the Aerodynamics of Wind Turbines, ETSU, Harwell (pp. 4-1).

- [41]. Landgrebe, A. J. (1972). The wake geometry of a hovering helicopter rotor and its influence on rotor performance. *Journal of the American Helicopter Society*, 17(4), 3-15.
- [42]. Kocurek, J. D., & Tangler, J. L. (1977). A prescribed wake lifting surface hover performance analysis. *Journal of the American Helicopter Society*, 22(1), 24-35.
- [43]. Egolf, T. A., & Landgrebe, A. J. (1983). Helicopter rotor wake geometry and its influence in forward flight. Volume 1: Generalized wake geometry and wake effect on rotor airloads and performance.
- [44]. Kocurek, D. (1987). Lifting surface performance analysis for horizontal axis wind turbines (No. SERI/STR-217-3163). Solar Energy Research Inst., Golden, CO (USA).
- [45]. Dumitrescu, H., & Cardos, V. (1998). Wind turbine aerodynamic performance by lifting line method. *International Journal of Rotating Machinery*, 4(3), 141-149.
- [46]. Coton, F. N., & Wang, T. (1999). The prediction of horizontal axis wind turbine performance in yawed flow using an unsteady prescribed wake model. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 213(1), 33-43.
- [47]. Anderson Jr, J. D. (2010). *Fundamentals of aerodynamics: fifth edition*. (5 ed.). Tata McGraw-Hill Education.
- [48]. Sant, T. (2007). Improving BEM-based aerodynamic models in wind turbine design codes. TU Delft, Delft University of Technology.
- [49]. Bottasso, C. L., Cacciola, S., & Iriarte, X. (2014). Calibration of wind turbine lifting line models from rotor loads. *Journal of Wind Engineering and Industrial Aerodynamics*, 124, 29-45
- [50]. Whale, J., Fisichella, C. J., & Selig, M. S. (2000). Correcting inflow measurements from wind turbines using a lifting-surface code. *Journal of solar energy engineering*, 122(4), 196-202
- [51]. Katz, J., & Plotkin, A. (2001). *Low-speed aerodynamics* (Vol. 13). Cambridge University Press.
- [52]. Roura, M., Cuerva, A., Sanz-Andrés, A., & Barrero-Gil, A. (2010). A panel method free-wake code for aeroelastic rotor predictions. *Wind Energy*, 13(4), 357-371.
- [53]. Hu, H., Gu, B., Zhang, H., Song, X., & Zhao, W. (2015). Hybrid Vortex Method for the Aerodynamic Analysis of Wind Turbine. *International Journal of Aerospace Engineering*.
- [54]. Simoes, F. J., & Graham, J. M. R. (1992). Application of a free vortex wake model to a horizontal axis wind turbine. *Journal of Wind Engineering and Industrial Aerodynamics*, 39(1), 129-138.

- [55]. Bareiss, R., & Wagner, S. (1993). The Free Wake/Hybrid Wake Code RVOLM—A Tool for Aerodynamic Analysis of Wind Turbines. In Proceedings of the European Community Wind Energy Conference (pp. 424-427).
- [56]. Sebastian, T., & Lackner, M. A. (2012). Development of a free vortex wake method code for offshore floating wind turbines. *Renewable Energy*, 46, 269-275.
- [57]. Currin, H. D., Coton, F. N., & Wood, B. (2008). Dynamic prescribed vortex wake model for AERODYN/FAST. *Journal of Solar Energy Engineering*, 130(3), 031007
- [58]. Sant, T., van Kuik, G., & van Bussel, G. J. W. (2006). Estimating the angle of attack from blade pressure measurements on the NREL phase VI rotor using a free wake vortex model: Axial conditions. *Wind Energy*, 9(6), 549-577.
- [59]. Schepers, J. G., & van Rooij, R. P. J. O. M. (2008). Analysis of aerodynamic measurements on a model wind turbine placed in the NASA-Ames tunnel. Energy Research Centre of the Netherlands ECN.
- [60]. Tangler, J., & Kocurek, J. D. (2005). Wind turbine post-stall airfoil performance characteristics guidelines for blade-element momentum methods. In 43rd AIAA Aerospace Sciences Meeting and Exhibit (pp. 10-13). Reno, Nevada.
- [61]. Breton, S. P., Coton, F. N., & Moe, G. (2008). A study on rotational effects and different stall delay models using a prescribed wake vortex scheme and NREL phase VI experiment data. *Wind Energy*, 11(5), 459-482.
- [62]. Wu, T. Y. (1962). Flow through a heavily loaded actuator disc. *Schiffstechnik*, 9(47), 134-138.
- [63]. Hough, G. R., & Ordway, D. E. (1964). The generalized actuator disk (No. TAR-TR-6401). THERM ADVANCED RESEARCH INC ITHACA NY.
- [64]. Greenberg, M. D. (1972). Nonlinear actuator disk theory. *Zeitschrift fuer Flugwissenschaften*, 20, 90-98.
- [65]. Sørensen, J. N., & Myken, A. (1992). Unsteady actuator disc model for horizontal axis wind turbines. *Journal of Wind Engineering and Industrial Aerodynamics*, 39(1-3), 139-149.
- [66]. Conway, J. T. (1995). Analytical solutions for the actuator disk with variable radial distribution of load. *Journal of Fluid Mechanics*, 297, 327-355.
- [67]. Sørensen, J. N., & Kock, C. W. (1995). A model for unsteady rotor aerodynamics. *Journal of wind engineering and industrial aerodynamics*, 58(3), 259-275.
- [68]. Madsen, H. A. (1997). A CFD analysis of the actuator disc flow compared with momentum theory results. Iea Joint Action. Aerodynamics of Wind Turbines.
- [69]. Conway, J. T. (1998). Exact actuator disk solutions for non-uniform heavy loading and slipstream contraction. *Journal of Fluid Mechanics*, 365, 235-267.

- [70]. Sørensen, J. N., & Shen, W. Z. (2002). Numerical modeling of wind turbine wakes. *Journal of fluids engineering*, 124(2), 393-399.
- [71]. Troldborg, N., Sørensen, J. N., & Mikkelsen, R. (2007). Actuator line simulation of wake of wind turbine operating in turbulent inflow. In *Journal of physics: conference series* (Vol. 75, No. 1, p. 012063). IOP Publishing
- [72]. Troldborg, N. (2009). Actuator line modeling of wind turbine wakes. PhD thesis, Technical University of Denmark
- [73]. Shen, W. Z., Sørensen, J. N., & Zhang, J. (2007). Actuator surface model for wind turbine flow computations. In *2007 European Wind Energy Conference and Exhibition*
- [74]. Shen, W. Z., Zhang, J. H., & Sørensen, J. N. (2009). The actuator surface model: a new Navier–Stokes based model for rotor computations. *Journal of Solar Energy Engineering*, 131(1), 011002.
- [75]. Dobrev, I., Massouh, F., & Rapin, M. (2007). Actuator surface hybrid model. In *Journal of Physics: Conference Series* (Vol. 75, No. 1, p. 012019). IOP Publishing.
- [76]. Sanderse, B., Pijl, S. P., & Koren, B. (2011). Review of computational fluid dynamics for wind turbine wake aerodynamics. *Wind Energy*, 14(7), 799-819.
- [77]. Carcangiu, C. E., Sørensen, J. N., Cambuli, F., & Mandas, N. (2007). CFD–RANS analysis of the rotational effects on the boundary layer of wind turbine blades. In *Journal of physics: conference series* (Vol. 75, No. 1, p. 012031). IOP Publishing.
- [78]. Jimenez, A., Crespo, A., Migoya, E., & Garcia, J. (2007). Advances in large-eddy simulation of a wind turbine wake. In *Journal of Physics: Conference Series* (Vol. 75, No. 1, p. 012041). IOP Publishing.
- [79]. Calaf, M., Meneveau, C., & Meyers, J. (2010). Large eddy simulation study of fully developed wind-turbine array boundary layers. *Physics of Fluids*, 22(1), 015110.
- [80]. Jiménez, Á., Crespo, A., & Migoya, E. (2010). Application of a LES technique to characterize the wake deflection of a wind turbine in yaw. *Wind energy*, 13(6), 559-572.
- [81]. Jensen, N. O. (1983). A note on wind generator interaction. Risø-M-2411, Risø National Laboratory.
- [82]. Katic, I., Højstrup, J., & Jensen, N. O. (1986). A simple model for cluster efficiency. In *European Wind Energy Association Conference and Exhibition* (pp. 407-410).
- [83]. Thørgersen, M. L., Sørensen, T., Nielsen, P., Grötzner, A., & Chun, S. (2005). WindPRO/PARK: introduction to wind turbine wake modelling and wake generated turbulence. EMD International A/S, Aalborg, Denmark.

- [84]. Mortensen, N. G., Heathfield, D. N., Myllerup, L., Landberg, L., & Rathmann, O. (2005). Wind atlas analysis and application program: WAsP 8 help facility (online). Risø National Laboratory, Roskilde, Denmark. ISBN 87-550-3457-8.2005.
- [85]. Tian, L., Zhu, W., Shen, W., Zhao, N., & Shen, Z. (2015). Development and validation of a new two-dimensional wake model for wind turbine wakes. *Journal of Wind Engineering and Industrial Aerodynamics*, 137, 90-99.
- [86]. Larsen, G. C. (1988). A simple wake calculation procedure. Risø-M-2760, Risø National Lab.
- [87]. Larsen, G. C., Højstrup, J., & Aagaard Madsen, H. (1996). Wind fields in wakes. In A. Zervos, H. Ehmann, & P. Helm (Eds.), *1996 European Union wind energy conference. Proceedings*. (pp. 764-768). Bedford: H.S. Stephens & Associates.
- [88]. Pierik, J. T. G., Dekker, J. W. M., Braam, H., Bulder, B. H., Winkelaar, D., Larsen, G. C., Chaviaropoulos, P., Derrick, A. & Molly, J. P. (1999). European wind turbine standards II (EWTS-II). *Wind Energy for the Next Millennium. Proceedings*.
- [89]. Frandsen, S., Barthelmie, R., Pryor, S., Rathmann, O., Larsen, S., Højstrup, J., & Thøgersen, M. (2006). Analytical modelling of wind speed deficit in large offshore wind farms. *Wind energy*, 9(1-2), 39-53.
- [90]. Barthelmie, R. J., Larsen, G. C., Frandsen, S. T., Folkerts, L., Rados, K., Pryor, S. C., Lange, B., & Schepers, G. (2006). Comparison of wake model simulations with offshore wind turbine wake profiles measured by sodar. *Journal of atmospheric and oceanic technology*, 23(7), 888-901.
- [91]. Ishihara, T., Yamaguchi, A., & Fujino, Y. (2004). Development of a new wake model based on a wind tunnel experiment. *Global wind power*, 1.
- [92]. Prospathopoulos, J. M., Politis, E. S., & Chaviaropoulos, P. K. (2008). Modelling wind turbine wakes in complex terrain. *Proceedings EWEK 2008*, Brussels, Belgium.
- [93]. Ainslie, J. F. (1988). Calculating the flowfield in the wake of wind turbines. *Journal of Wind Engineering and Industrial Aerodynamics*, 27(1), 213-224
- [94]. Lange, B., Waldl, H. P., Guerrero, A. G., Heinemann, D., & Barthelmie, R. J. (2003). Modelling of offshore wind turbine wakes with the wind farm program FLaP. *Wind Energy*, 6(1), 87-104.
- [95]. Crespo, A., Hernandez, J., Fraga, E., & Andreu, C. (1988). Experimental validation of the UPM computer code to calculate wind turbine wakes and comparison with other models. *Journal of Wind Engineering and Industrial Aerodynamics*, 27(1-3), 77-88

- [96]. Crespo, A., & Hernández, J. (1989). Numerical modelling of the flow field in a wind turbine wake. In Proceedings of the 3rd Joint ASCE/ASME Mechanics Conference (pp. 121-127).
- [97]. Schepers, J. G. (2003). ENDOW: Validation and improvement of ECN's wake model (p. 113). Energy research Centre of the Netherlands ECN.
- [98]. Malki, R., Masters, I., Williams, A. J., & Croft, T. N. (2014). Planning tidal stream turbine array layouts using a coupled blade element momentum–computational fluid dynamics model. *Renewable Energy*, 63, 46-54.
- [99]. Bartl, J., Pierella, F., & Sætrana, L. (2012). Wake measurements behind an array of two model wind turbines. *Energy Procedia*, 24, 305-312.
- [100]. Quarton, D., & Ainslie, J. (1990). Turbulence in wind turbine wakes. *Wind Engineering*, 14(1), 15-23.
- [101]. Vermeulen, P. E. J. (1980). An experimental analysis of wind turbine wakes. In 3rd international symposium on wind energy systems (Vol. 1, pp. 431-450).
- [102]. Hassan, U. (1993). A wind tunnel investigation of the wake structure within small wind turbine farms. Harwell Laboratory, Energy Technology Support Unit.
- [103]. Frandsen, S. T., & Thøgersen, M. L. (1999). Integrated fatigue loading for wind turbines in wind farms by combining ambient turbulence and wakes. *Wind Engineering*.
- [104]. Voutsinas, S. G., Rados, K. G., & Zervos, A. (1992). On the effect of the rotor geometry on the formation and the development of its wake. *Journal of Wind Engineering and Industrial Aerodynamics*, 39(1), 283-291.
- [105]. Voutsinas, S. G., Glekas, J. P., & Zervos, A. (1992). Investigation of the effect of the initial velocity profile on the wake development of a wind turbine. *Journal of Wind Engineering and Industrial Aerodynamics*, 39(1-3), 293-301.

CHAPTER 4

- [1]. Wilcox, D. C. (1998). *Turbulence modeling for CFD*. La Canada, CA: DCW industries.
- [2]. Durbin, P. A. (1991). Near-wall turbulence closure modeling without “damping functions”. *Theoretical and Computational Fluid Dynamics*, 3(1), 1-13.
- [3]. Launder, B. E., Reece, G. J., & Rodi, W. (1975). Progress in the development of a Reynolds-stress turbulence closure. *Journal of Fluid Mechanics*, 68(03), 537-566.
- [4]. Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8), 1598-1605.

- [5]. Ansys Fluent. (2009). Chapter 4: in ANSYS FLUENT 12.0 Theory Guide. ANSYS Inc.
- [6]. Ansys Fluent. (2009). Chapter 2: in ANSYS FLUENT 12.0 Theory Guide. ANSYS Inc.
- [7]. Ansys Fluent. (2009). Moving zones: in introductory FLUENT TRAINING. ANSYS Inc.
- [8]. Ansys Fluent. (2009). Chapter 3: in ANSYS FLUENT 12.0 Theory Guide. ANSYS Inc.

CHAPTER 5

- [1]. Hand, M. M., Simms, D. A., Fingersh, L. J., Jager, D. W., Cotrell, J. R., Schreck, S., & Larwood, S. M. (2001). Unsteady aerodynamics experiment phase VI: wind tunnel test configurations and available data campaigns. National Renewable Energy Laboratory, Golden, CO, Report No. NREL/TP-500-29955.
- [2]. Schreck, S. (2002). The NREL full-scale wind tunnel experiment Introduction to the special issue. *Wind Energy*, 5(2-3), 77-84.
- [3]. Ansys Fluent. (2009). Turbulence modeling: in introductory FLUENT TRAINING. ANSYS Inc.
- [4]. Simms, D., Schreck, S., Hand, 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, Report No. NREL/TP-500-29494.
- [5]. Blocken, B., Stathopoulos, T., & Carmeliet, J. (2007). CFD simulation of the atmospheric boundary layer: wall function problems. *Atmospheric environment*, 41(2), 238-252.
- [6]. Cabezón, D., Migoya, E., & Crespo, A. (2011). Comparison of turbulence models for the computational fluid dynamics simulation of wind turbine wakes in the atmospheric boundary layer. *Wind Energy*, 14(7), 909-921.
- [7]. Zhang, X. (2009). CFD simulation of neutral ABL flows. Danmarks Tekniske Universitet, Risø Nationallaboratoriet for Bæredygtig Energi.

CHAPTER 6

- [1]. Sørensen, N. N., Michelsen, J. A., & Schreck, S. (2002). Navier – Stokes predictions of the NREL phase VI rotor in the NASA Ames 80 ft × 120 ft wind tunnel. *Wind Energy*, 5(2 - 3), 151-169.

- [2]. Li, Y., Paik, K. J., Xing, T., & Carrica, P. M. (2012). Dynamic overset CFD simulations of wind turbine aerodynamics. *Renewable Energy*, 37(1), 285-298.
- [3]. Johansen, J., & Sørensen, N. N. (2004). Aerofoil characteristics from 3D CFD rotor computations. *Wind Energy*, 7(4), 283-294
- [4]. Tangler, J., & Kocurek, J. D. (2005). Wind turbine post-stall airfoil performance characteristics guidelines for blade-element momentum methods. In 43rd AIAA Aerospace Sciences Meeting and Exhibit (pp. 10-13). Reno, Nevada
- [5]. Sant, T., van Kuik, G., & van Bussel, G. J. W. (2006). Estimating the angle of attack from blade pressure measurements on the NREL phase VI rotor using a free wake vortex model: Axial conditions. *Wind Energy*, 9(6), 549-577.
- [6]. Schepers, J. G., & van Rooij, R. P. J. O. M. (2008). Analysis of aerodynamic measurements on a model wind turbine placed in the NASA-Ames tunnel. Energy Research Centre of the Netherlands ECN.
- [7]. Viterna, L. A., & Corrigan, R. D. (1982). Fixed pitch rotor performance of large horizontal axis wind turbines. DOE/NASA Workshop on Large Horizontal Axis Wind Turbines, Cleveland, OH, 1981; 69–85.
- [8]. Lanzafame, R., & Messina, M. (2007). Fluid dynamics wind turbine design: Critical analysis, optimization and application of BEM theory. *Renewable energy*, 32(14), 2291-2305.
- [9]. Tangler, J., & Kocurek, J. D. (2005). Wind turbine post-stall airfoil performance characteristics guidelines for blade-element momentum methods. In 43rd AIAA Aerospace Sciences Meeting and Exhibit (pp. 10-13). Reno, Nevada.
- [10]. Snel, H., Houwink, R., & Bosschers, J. (1994). Sectional prediction of lift coefficients on rotating wind turbine blades in stall. Petten, Netherlands: Netherlands Energy Research Foundation
- [11]. Du, Z., & Selig, M. (1998). A 3-D stall-delay model for horizontal axis wind turbine performance prediction. In 1998 ASME Wind Energy Symposium (p. 21).
- [12]. Chaviaropoulos, P., & Hansen, M. (2000). Investigating three-dimensional and rotational effects on wind turbine blades by means of a quasi-3D Navier-Stokes solver. *Journal of fluids engineering*, 122(2), 330-336.
- [13]. Lindenburg, C. (2004). Modelling of rotational augmentation based on engineering considerations and measurements. In European Wind Energy Conference (EWEC 2004), London, UK, Nov (pp. 22-25).
- [14]. Jensen, N. O. (1983). A note on wind generator interaction. Risø-M-2411, Risø National Laboratory.
- [15]. Larsen, G. C. (1988). A simple wake calculation procedure. Risø-M-2760, Risø National Lab.

- [16]. Frandsen, S., Barthelmie, R., Pryor, S., Rathmann, O., Larsen, S., Højstrup, J., & Thøgersen, M. (2006). Analytical modelling of wind speed deficit in large offshore wind farms. *Wind energy*, 9(1-2), 39-53.
- [17]. Ishihara, T., Yamaguchi, A., & Fujino, Y. (2004). Development of a new wake model based on a wind tunnel experiment. *Global wind power*, 1.
- [18]. Quarton, D., & Ainslie, J. (1990). Turbulence in wind turbine wakes. *Wind Engineering*, 14(1), 15-23.
- [19]. Hassan, U. (1993). A wind tunnel investigation of the wake structure within small wind turbine farms. Harwell Laboratory, Energy Technology Support Unit.
- [20]. Larsen, G. C., Højstrup, J., & Aagaard Madsen, H. (1996). Wind fields in wakes. In A. Zervos, H. Ehmann, & P. Helm (Eds.), 1996 European Union wind energy conference. Proceedings. (pp. 764-768). Bedford: H.S. Stephens & Associates.
- [21]. Chamorro, L. P., & Porté-Agel, F. (2009). A wind-tunnel investigation of wind-turbine wakes: boundary-layer turbulence effects. *Boundary-layer meteorology*, 132(1), 129-149.

CHAPTER 7

- [1]. Javaherchi, T., Antheaume, S., & Aliseda, A. (2014). Hierarchical methodology for the numerical simulation of the flow field around and in the wake of horizontal axis wind turbines: Rotating reference frame, blade element method and actuator disk model. *Wind Engineering*, 38(2), 181-201.
- [2]. Sturge, D., Sobotta, D., Howell, R., While, A., & Lou, J. (2015). A hybrid actuator disc–Full rotor CFD methodology for modelling the effects of wind turbine wake interactions on performance. *Renewable Energy*, 80, 525-537.
- [3]. Ameer, K., Masson, C., & Eecen, P. J. (2011). 2D and 3D numerical simulation of the wind-rotor/nacelle interaction in an atmospheric boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics*, 99(8), 833-844
- [4]. Phillips, A. B., Turnock, S. R., & Furlong, M. (2010). Accurate capture of propeller-rudder interaction using a coupled blade element momentum-rans approach. *Ship Technology Research*, 57(2), 128-139.
- [5]. Masters, I., Malki, R., Williams, A. J., & Croft, T. N. (2013). The influence of flow acceleration on tidal stream turbine wake dynamics: A numerical study using a coupled BEM–CFD model. *Applied Mathematical Modelling*, 37(16), 7905-7918.
- [6]. Sørensen, J. N., & Shen, W. Z. (2002). Numerical modeling of wind turbine wakes. *Journal of fluids engineering*, 124(2), 393-399.

- [7]. Shen, W. Z., Sørensen, J. N., & Zhang, J. (2007). Actuator surface model for wind turbine flow computations. In 2007 European Wind Energy Conference and Exhibition.
- [8]. Dobrev, I., Massouh, F., & Rapin, M. (2007). Actuator surface hybrid model. In Journal of Physics: Conference Series (Vol. 75, No. 1, p. 012019). IOP Publishing.

Appendix A: BEM MATLAB code (Buhl's Correction Factor and Lazafame and Messina's model of aerofoil characteristics extrapolation)

```

clear all
% a-axial induction factor
% a_dash-angular induction factor
% alpha-angle of attack
% chord-chord length
% C_normal-elemental normal factor
% C_torque-elemental torque factor
% C_drag-drag coefficient
% C_lift-lift coefficient
% C_power-power coefficient
% dr-radial width of each blade element
% Dthrust-thrust on each blade element
% Dtorque-torque on each blade element
% lambda-tip speed ratio (TSR)
% lamda_r-local tip speed ratio
% N-number of blade elements
% phi-inflow angle between relative velocity and plane of rotation
% Re-Reynolds number
% R-total length of the blade(m)
% hubradius- radius of hub or root(m)
% lengths are normalized by the radius of tip
% angles are in degrees except phi
% radius(i)-radius of the midpoint of the ith element
% chord(i) - corresponding chord of ith element at radius(i)
% twist(i) - corresponding twist of ith element at radius(i)
% sigma - local solidity,
% U_rel - effective velocity at blade element
% U_wind - wind speed (m/s)
% Enter the datas of the wind turbine blades
Blades=input(' Number of blades:');
Bladepitchangle=input(' Bladepitchangle:');
NofD= input(' Number of datas:');
R= input(' Radius of tip in metre:');
load NRELPHASEVIbladegeometrydata.dat;
given_R=NRELPHASEVIbladegeometrydata(:,1);
given_C=NRELPHASEVIbladegeometrydata(:,2);
given_twist=NRELPHASEVIbladegeometrydata(:,3);
for i=1:NofD
    given_r(i)=given_R(i)/R;
end
for i=1:NofD
    given_c(i)=given_C(i)/R;
end
for i=1:NofD
    twist_datas(i)=given_twist(i);
end
%Fit a 4th order polynomial for chord and radius and the coefficient is
as follow
coeffrvsch=polyfit(given_r,given_c,4);
%Fit a 10th order polynomial for twist and radius and the coefficient is
as follow
coeffrvst=polyfit(given_r,twist_datas,10);
N= input('Enter number of blade elements:');
hubradius= input('Enter radius of root in metre:');
hub = hubradius/R;
dr = ( 1.0 - hub)/N;

```

```

for i = 1:N
    if (i == 1)
        radius(1) = hub + dr/2;
    else
        radius(i) = radius(i-1) + dr;
    end
    chord(i) =
coeffrvsch(1)*radius(i)^4+coeffrvsch(2)*radius(i)^3+coeffrvsch(3)*radius
(i)^2+coeffrvsch(4)*radius(i)+coeffrvsch(5);
    twist(i) = coeffrvst(1)*radius(i)^10+coeffrvst(2)*
radius(i)^9+coeffrvst(3)* radius(i)^8+coeffrvst(4)*
radius(i)^7+coeffrvst(5)* radius(i)^6+coeffrvst(6)*
radius(i)^5+coeffrvst(7)* radius(i)^4+coeffrvst(8)*
radius(i)^3+coeffrvst(9)* radius(i)^2+ coeffrvst(10)*
radius(i)+coeffrvst(11);
end
% Kinematic viscosity of air
viscosity = input('Enter the value of viscosity in m^2/S:');
% Density of air
density = input('Enter the value of density in Kg/m^3:');
% tolerance
tolerance = 0.001;
U_wind = input(' Enter the wind speed in m/s: ');
%Rotor rotational speed in RPM for NREL Phase VI Turbine is 72 rpm%
%Angular Velocity of rotor in rad/s
omega=7.5398223599963545;
lamda=(omega*R)/U_wind;
TotalThrust = 0.0;
TotalTorque = 0.0;
fprintf('\n')
fprintf(' radius(r/R)   radiusinmetre   iteration   alpha
a      U_wind*(1-a)    C_lift          C_drag      Dtorque
Re  \n')
for i = 1: N
% Initialise a and a'
    a = 0.0;
    a_dash = 0.0;
    Dtorque = 0.0;
% Local speed ratio
    lamda_r = lamda*radius(i);
% Solidity
    sigma=Blades*chord(i)/(2*pi*radius(i));
    diffbtwna = 300*tolerance;
    j=1;% j is used to find the number of iterations
    while diffbtwna > tolerance
        j=j+1;
        phi = atan((1 - a)/((1 + a_dash)*lamda_r)) ;
        cosphi = cos(phi);
        sinphi = sin(phi);
        alpha = (phi*180.0/pi) - twist(i)-Bladepitchangle;
%Axial velocity just just infront of turbine relative to windspeed
relative to wind speed
        v=(1-a);
%circumferential velocity of the rotor relative to windpseed
        u=(lamda_r*(1+a_dash));
        U_rel = sqrt(v^2 + u^2);
        Re = U_rel*U_wind*chord(i)*R/viscosity ;
% Find C_lift and C_drag
        if alpha<=20
            load clandcdforaobetweenminus6to20.dat;
            AOAxis=clandcdforaobetweenminus6to20(:,1);

```

```

CLyaxis=clandcdforaoabetweenminus6to20(:,2);
CDyaxis=clandcdforaoabetweenminus6to20(:,3);
%To find CL
%8th order polynomial fit between Angle of Attack and Coefficient of
lift(has to get best fitcurve)
AOAvsCL=polyfit(AOAxaxis, CLyaxis,7);
C_lift= AOAvsCL(1)*alpha^7+ AOAvsCL(2)*alpha^6+ AOAvsCL(3)*alpha^5+
AOAvsCL(4)*alpha^4+ AOAvsCL(5)*alpha^3+ AOAvsCL(6)*alpha^2+
AOAvsCL(7)*alpha^1+AOAvsCL(8);
%To find drag
%Exponential fit between Angle of Attack and Coefficient of drag(has to
get best fitcurve)
AOAvsCD=fit(AOAxaxis,CDyaxis,'exp2');
C_drag=AOAvsCD(alpha);
else
%For flat plate fully stalled region(20<alpha<45)
%CLmax is CL at alpha 45degree and CD max is CD at alpha 90 degree
%From experimental CLmax and CDmax
CLmax=1.12;
CDmax=2.27;
C_lift=2*CLmax *sind(alpha)*cosd(alpha);
C_drag= CDmax* sind(alpha)* sind(alpha);
end
C_normal = C_lift*cosphi + C_drag*sinphi;
C_torque = C_lift*sinphi - C_drag*cosphi;
%Prandtl Tip Loss Correction Factor
f_tip=0.5*Blades*((1/radius(i))-1)/sin(phi);
F_tip=2*acos(exp(-f_tip))/pi;
%Hub Loss Factor
f_hub=0.5*Blades*(1-(hub/radius(i)))/sinphi;
F_hub=2*acos(exp(-f_hub))/pi;
F=F_tip*F_hub;
%Buhl Model
C_T=4*F*a*(1-a);
if C_T<(0.96*F)
a_new = 1/(((4*F*(sinphi)^2)/(sigma*C_normal))+1);
else
a_new=((18*F)-20-(3*((C_T*(50-(36*F)))+(12*F*((3*F)-4)))^0.5))/((36*F)-
50);
end
diffbtwna = abs(a - a_new);
a = 0.5*(a + a_new); % Average old and new a
a_dash =1/(((4* F*sinphi*cosphi)/(sigma*C_torque))-
1);
end
%Thrust and torque is multiplied U_wind^2 and R^2
because normalization of U_rel with U_wind and chord ,radius and dr with
radius of the tip
Dthrust =
0.5*density*Blades*U_rel*U_rel*chord(i)*C_normal*dr*U_wind*U_wind*R*R;
Dtorque =
0.5*density*Blades*U_rel*U_rel*chord(i)*C_torque*radius(i)*dr*U_wind*U_
ind*R*R*R;
TotalThrust = TotalThrust + Dthrust;
TotalTorque = TotalTorque + Dtorque;
radiusinmetre=radius(i)*R;
Numberofiteration=j;
Velocityatrotor=U_wind*(1-a);

```

```

fprintf(' %7.5f %7.5f %2d %7.3f %7.4f %7.4f
%7.4f %8.5f %8.6f %8.4e \n',
radius(i),radiusinmetre ,Numberofiteration , alpha, a,Velocityatrotor,
C_lift, C_drag, Dtorque, Re)
end
% Power in Watts
power = TotalTorque*omega;
fprintf(' Power = %5e Watts, TotalThrust = %5e Newtons
\n',power,TotalThrust)
fprintf(' \n')
% power coefficient
C_power = power/(0.5*density*(U_wind^3)*pi*R^2);
C_thrust= TotalThrust/(0.5*density*(U_wind^2)*pi*R^2);
fprintf(' \n')
fprintf(' C_power = %5f, C_thrust = %5f \n' , C_power, C_thrust)

```

APPENDIX B: Inverse BEM MATLAB code

```
Blades=input(' Number of blades:');
Bladepitchangle=input(' Bladepitchangle:');
NofD= input(' Number of datas:');
R= input(' Radius of tip in metre:');
hubradius= input('Enter radius of root in metre:');
load NRELPHASEVIbladegeometrydata.dat;
given_R=NRELPHASEVIbladegeometrydata(:,1);
given_C=NRELPHASEVIbladegeometrydata(:,2);
given_twist=NRELPHASEVIbladegeometrydata(:,3);
given_twist_axis= NRELPHASEVIbladegeometrydata(:,4);
for i=1:NofD
    radius(i)=given_R(i);
end
for i=1:NofD
    chord_length(i)=given_C(i);
end
for i=1:NofD
    twist(i)=given_twist(i);
end
for i=1:NofD
    twistaxis(i)=given_twist_axis(i);
end
tolerence = 0.001;
diffbtwna = 300*tolerance;
U_wind = input(' Enter the wind speed in m/s: ');
omega=7.5398223599963545;
density=1.225;
Pr_atm=input('Atmospheric Pressure:');
j=1;
for j=1:NofD
    if j==1
        importl=xlsread('PressureValues1.xls'); % Read excel file
        x1=importl(:,1);
        y1=importl(:,2);
        z1=importl(:,3);
        Pressure1=importl(:,4);
        twist_angle1=twist(j)+Bladepitchangle;
        % Distance between the leading edge and twist axis %
        twist_axis1=twistaxis(j);
        chord1=chord_length(j);
        x_rotated1=(x1*cosd(twist_angle1))+(y1*sind(twist_angle1));
        y_rotated1=(-x1*sind(twist_angle1))+(y1*cosd(twist_angle1));
        z_rotated1=z1;
        x_translated1=x_rotated1+twist_axis1;
        y_translated1=y_rotated1;
        z_translated1=z_rotated1;
        Cp1=(Pressure1-
        Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated1(1,:)).^2));
        NegCp1=-1*Cp1(:,1);
        xbyc1=x_translated1/chord1;
        ybyc1=y_translated1/chord1;
        % Initialize row index for upper and lower surfaces
        urow1=1;
        lrow1=1;
        % Initialize upper and lower surface arrays
        usurf1=[];
        lsurf1=[];
        i=1;
        for i=1:size(importl,1)
```



```

        if y_translated1(i)>0.0000000
            usurf1(urow1,:)=[x_translated1(i) y_translated1(i) xbyc1(i)
ybyc1(i) Cp1(i) NegCp1(i)];
            urow1=urow1+1;
        else
            lsurf1(lrow1,:)=[x_translated1(i) y_translated1(i) xbyc1(i)
ybyc1(i) Cp1(i) NegCp1(i)];
            lrow1=lrow1+1;
        end
    end
    usurface1=sortrows(usurf1,3);
    lsurface1=sortrows(lsurf1,3);
    Cn1=-
    trapz(usurface1(:,3),usurface1(:,5))+trapz(lsurface1(:,3),lsurface1(:,5)
);
    Ct1=-
    trapz(lsurface1(:,4),lsurface1(:,5))+trapz(usurface1(:,4),usurface1(:,5)
);
    Ct1=-1*Ct1;
    Cthrust1=(Cn1*cosd(twist_angle1))-(Ct1*sind(twist_angle1));
    Ctorque1=(Cn1*sind(twist_angle1))+(Ct1*cosd(twist_angle1));
    tolerance=1e-5;
    a=0;
    a_dash=0;
    a_new=0;
    diff_btwn_a_and_a_anew=300*tolerance;
    while diff_btwn_a_and_a_anew>tolerance
        phil=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated1(1,:)));
        f_tip=0.5*Blades*(R-
z_translated1(1,:)/(z_translated1(1,:)*sin(phil)));
        F_tip=(2/pi)*acos(exp(-1*f_tip));
        f_hub=0.5*Blades*((z_translated1(1,:)-
hubradius)/(z_translated1(1,:)*sin(phil)));
        F_hub=(2/pi)*acos(exp(-1*f_hub));
        F=F_tip*F_hub;
        %Buhl Model
        if a<=0.4

a_new=1/(((8*pi*z_translated1(1,:)*F*sin(phil)*sin(phil))/(chord1*Blades
*Cthrust1))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-
4)))^0.5)/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated1(1,:)*F*sin(phil)*cos(phil))/(chord1*Blade
s*Ctorque1))-1);
    end
    CL1=(Cthrust1*cos(phil))+(Ctorque1*sin(phil));
    CD1=(Cthrust1*sin(phil))-(Ctorque1*cos(phil));
    AoA1=(phil*180/pi)-twist_angle1;
    elseif j==2
        import2=xlsread('PressureValues2.xls'); % Read excel file
        x2=import2(:,1);
        y2=import2(:,2);
        z2=import2(:,3);
        Pressure2=import2(:,4);
        twist_angle2=twist(j)+Bladepitchangle;

```

```

% Distance between the leading edge and twist axis %
twist_axis2=twistaxis(j);
chord2=chord_length(j);
x_rotated2=(x2*cosd(twist_angle2))+(y2*sind(twist_angle2));
y_rotated2=(-x2*sind(twist_angle2))+(y2*cosd(twist_angle2));
z_rotated2=z2;
x_translated2=x_rotated2+twist_axis2;
y_translated2=y_rotated2;
z_translated2=z_rotated2;
Cp2=(Pressure2-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated2(1,:)).^2));
NegCp2=-1*Cp2(:,1);
xbyc2=x_translated2/chord2;
ybyc2=y_translated2/chord2;
% Initialize row index for upper and lower surfaces
urow2=1;
lrow2=1;
% Initialize upper and lower surface arrays
usurf2=[];
lsurf2=[];
i=1;
for i=1:size(import2,1)
    if y_translated2(i)>0.0000000
        usurf2(urow2,:)=[x_translated2(i) y_translated2(i) xbyc2(i)
ybyc2(i) Cp2(i) NegCp2(i)];
        urow2=urow2+1;
    else
        lsurf2(lrow2,:)=[x_translated2(i) y_translated2(i) xbyc2(i)
ybyc2(i) Cp2(i) NegCp2(i)];
        lrow2=lrow2+1;
    end
end
usurface2=sortrows(usurf2,3);
lsurface2=sortrows(lsurf2,3);
Cn2=-
trapz(usurface2(:,3),usurface2(:,5))+trapz(lsurface2(:,3),lsurface2(:,5)
);
Ct2=-
trapz(lsurface2(:,4),lsurface2(:,5))+trapz(usurface2(:,4),usurface2(:,5)
);
Ct2=-1*Ct2;
Cthrust2=(Cn2*cosd(twist_angle2))-(Ct2*sind(twist_angle2));
Ctorque2=(Cn2*sind(twist_angle2))+(Ct2*cosd(twist_angle2));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phi2=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated2(1,:)));
    f_tip=0.5*Blades*(R-
z_translated2(1,:))/(z_translated2(1,:)*sin(phi2));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated2(1,:)-
hubradius)/(z_translated2(1,:)*sin(phi2)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

```

```

a_new=1/(((8*pi*z_translated2(1,:)*F*sin(phi2)*sin(phi2))/(chord2*Blades
*Cthrust2))+1);
    else
        C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
        a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-
4))))^0.5)/((36*F)-50);
    end
    diff_btwn_a_and_a_anew=abs(a-a_new);
    a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated2(1,:)*F*sin(phi2)*cos(phi2))/(chord2*Blade
s*Ctorque2))-1);
end
CL2=(Cthrust2*cos(phi2))+(Ctorque2*sin(phi2));
CD2=(Cthrust2*sin(phi2))-(Ctorque2*cos(phi2));
AoA2=(phi2*180/pi)-twist_angle2;
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phi2=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated2(1,:)));
    f_tip=0.5*Blades*((R-
z_translated2(1,:))/(z_translated2(1,:)*sin(phi2)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated2(1,:)-
hubradius)/(z_translated2(1,:)*sin(phi2)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated2(1,:)*F*sin(phi2)*sin(phi2))/(chord2*Blades
*Cthrust2))+1);
    else
        C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
        a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-
4))))^0.5)/((36*F)-50);
    end
    diff_btwn_a_and_a_anew=abs(a-a_new);
    a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated2(1,:)*F*sin(phi2)*cos(phi2))/(chord2*Blade
s*Ctorque2))-1);
end
CL2=(Cthrust2*cos(phi2))+(Ctorque2*sin(phi2));
CD2=(Cthrust2*sin(phi2))-(Ctorque2*cos(phi2));
AoA2=(phi2*180/pi)-twist_angle2;
elseif j==3
    import3=xlsread('PressureValues3.xls'); % Read excel file
    x3=import3(:,1);
    y3=import3(:,2);
    z3=import3(:,3);
    Pressure3=import3(:,4);
    twist_angle3=twist(j)+Bladepitchangle;
    % Distance between the leading edge and twist axis %
    twist_axis3=twistaxis(j);
    chord3=chord_length(j);
    x_rotated3=(x3*cosd(twist_angle3))+(y3*sind(twist_angle3));

```

```

y_rotated3=(-x3*sind(twist_angle3))+(y3*cosd(twist_angle3));
z_rotated3=z3;
x_translated3=x_rotated3+twist_axis3;
y_translated3=y_rotated3;
z_translated3=z_rotated3;
Cp3=(Pressure3-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated3(1,:)).^2));
NegCp3=-1*Cp3(:,1);
xbyc3=x_translated3/chord3;
ybyc3=y_translated3/chord3;
% Initialize row index for upper and lower surfaces
urow3=1;
lrow3=1;
% Initialize upper and lower surface arrays
usurf3=[];
lsurf3=[];
i=1;
for i=1:size(import3,1)
    if y_translated3(i)>0.0000000
        usurf3(urow3,:)= [x_translated3(i) y_translated3(i) xbyc3(i)
ybyc3(i) Cp3(i) NegCp3(i)];
        urow3=urow3+1;
    else
        lsurf3(lrow3,:)= [x_translated3(i) y_translated3(i) xbyc3(i)
ybyc3(i) Cp3(i) NegCp3(i)];
        lrow3=lrow3+1;
    end
end
usurface3=sortrows(usurf3,3);
lsurface3=sortrows(lsurf3,3);
subplot(3,2,1)
exp_30=xlsread('exp_30.xls');
exp_x_30=exp_30(:,1);
exp_Cp_30=exp_30(:,2);
lower_range=exp_30(:,5);
upper_range=exp_30(:,6);
width=3;
height=3;
axes_line_width=2;
font_size=8;
line_width=1.5;
marker_size=10;
pos=get(gcf, 'Position');
set(gcf, 'Position', [pos(1) pos(2) width*100 height*100]);
set(gca, 'FontSize', font_size, 'LineWidth', axes_line_width);
plot(xbyc3, NegCp3(:,1), 'blue --
', 'LineWidth', axes_line_width, 'MarkerSize', marker_size);
hold on
errorbar(exp_x_30, exp_Cp_30, lower_range, upper_range, 'red
*', 'LineWidth', axes_line_width, 'MarkerSize', marker_size);
xlim([-0.1, 1.1]);
ylim([-2, 8]);
xlabel('X/Chord');
ylabel('-Cp');
str='r/R';str='r/R';
prompt = 'r/R ';
str1 = input(prompt, 's');
title([str ':' str1]);
Cn3=-
trapz(usurface3(:,3),usurface3(:,5))+trapz(lsurface3(:,3),lsurface3(:,5)
);

```

```

Ct3=-
trapz(lsurface3(:,4),lsurface3(:,5))+trapz(usurface3(:,4),usurface3(:,5)
);
Ct3=-1*Ct3;
Cthrust3=(Cn3*cosd(twist_angle3))-(Ct3*sind(twist_angle3));
Ctorque3=(Cn3*sind(twist_angle3))+(Ct3*cosd(twist_angle3));
tolerence=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerence
    phi3=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated3(1,:)));
    f_tip=0.5*Blades*(R-
z_translated3(1,:)/(z_translated3(1,:)*sin(phi3)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated3(1,:)-
hubradius)/(z_translated3(1,:)*sin(phi3)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated3(1,:)*F*sin(phi3)*sin(phi3))/(chord3*Blades
*Cthrust3))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-
4)))^0.5))/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated3(1,:)*F*sin(phi3)*cos(phi3))/(chord3*Blade
s*Ctorque3))-1);
    end
    CL3=(Cthrust3*cos(phi3))+(Ctorque3*sin(phi3));
    CD3=(Cthrust3*sin(phi3))-(Ctorque3*cos(phi3));
    AoA3=(phi3*180/pi)-twist_angle3;
elseif j==4
    import4=xlsread('PressureValues4.xls'); % Read excel file
    x4=import4(:,1);
    y4=import4(:,2);
    z4=import4(:,3);
    Pressure4=import4(:,4);
    twist_angle4=twist(j)+Bladepitchangle;
    % Distance between the leading edge and twist axis %
    twist_axis4=twistaxis(j);
    chord4=chord_length(j);
    x_rotated4=(x4*cosd(twist_angle4))+(y4*sind(twist_angle4));
    y_rotated4=(-x4*sind(twist_angle4))+(y4*cosd(twist_angle4));
    z_rotated4=z4;
    x_translated4=x_rotated4+twist_axis4;
    y_translated4=y_rotated4;
    z_translated4=z_rotated4;
    Cp4=(Pressure4-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated4(1,:)).^2));
    NegCp4=-1*Cp4(:,1);
    xbyc4=x_translated4/chord4;
    ybyc4=y_translated4/chord4;
    % Initialize row index for upper and lower surfaces

```

```

urow4=1;
lrow4=1;
% Initialize upper and lower surface arrays
usurf4=[];
lsurf4=[];
i=1;
for i=1:size(import4,1)
    if y_translated4(i)>0.0000000
        usurf4(urow4,:)= [x_translated4(i) y_translated4(i) xbyc4(i)
ybyc4(i) Cp4(i) NegCp4(i)];
        urow4=urow4+1;
    else
        lsurf4(lrow4,:)= [x_translated4(i) y_translated4(i) xbyc4(i)
ybyc4(i) Cp4(i) NegCp4(i)];
        lrow4=lrow4+1;
    end
end
usurface4=sortrows(usurf4,3);
lsurface4=sortrows(lsurf4,3);
Cn4=-
trapz(usurface4(:,3),usurface4(:,5))+trapz(lsurface4(:,3),lsurface4(:,5)
);
Ct4=-
trapz(lsurface4(:,4),lsurface4(:,5))+trapz(usurface4(:,4),usurface4(:,5)
);
Ct4=-1*Ct4;
Cthrust4=(Cn4*cosd(twist_angle4)-(Ct4*sind(twist_angle4)));
Ctorque4=(Cn4*sind(twist_angle4)+(Ct4*cosd(twist_angle4)));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phi4=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated4(1,:)));
    f_tip=0.5*Blades*( (R-
z_translated4(1,:))/(z_translated4(1,:)*sin(phi4)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*( (z_translated4(1,:)-
hubradius)/(z_translated4(1,:)*sin(phi4)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated4(1,:)*F*sin(phi4)*sin(phi4))/(chord4*Blades
*Cthrust4))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*((C_T*(50-(36*F)))+(12*F*((3*F)-
4)))^0.5))/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated4(1,:)*F*sin(phi4)*cos(phi4))/(chord4*Blade
s*Ctorque4))-1);
    end
    Cl4=(Cthrust4*cos(phi4)+(Ctorque4*sin(phi4)));
    Cd4=(Cthrust4*sin(phi4)-(Ctorque4*cos(phi4)));
    AoA4=(phi4*180/pi)-twist_angle4;

```

```

elseif j==5
    import5=xlsread('PressureValues5.xls'); % Read excel file
    x5=import5(:,1);
    y5=import5(:,2);
    z5=import5(:,3);
    Pressure5=import5(:,4);
    twist_angle5=twist(j)+Bladepitchangle;
    % Distance between the leading edge and twist axis %
    twist_axis5=twistaxis(j);
    chord5=chord_length(j);
    x_rotated5=(x5*cosd(twist_angle5))+(y5*sind(twist_angle5));
    y_rotated5=(-x5*sind(twist_angle5))+(y5*cosd(twist_angle5));
    z_rotated5=z5;
    x_translated5=x_rotated5+twist_axis5;
    y_translated5=y_rotated5;
    z_translated5=z_rotated5;
    Cp5=(Pressure5-
    Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated5(1,:)).^2));
    NegCp5=-1*Cp5(:,1);
    xbyc5=x_translated5/chord5;
    ybyc5=y_translated5/chord5;
    % Initialize row index for upper and lower surfaces
    urow5=1;
    lrow5=1;
    % Initialize upper and lower surface arrays
    usurf5=[];
    lsurf5=[];
    i=1;
    for i=1:size(import5,1)
        if y_translated5(i)>0.0000000
            usurf5(urow5,:)= [x_translated5(i) y_translated5(i) xbyc5(i)
            ybyc5(i) Cp5(i) NegCp5(i)];
            urow5=urow5+1;
        else
            lsurf5(lrow5,:)= [x_translated5(i) y_translated5(i) xbyc5(i)
            ybyc5(i) Cp5(i) NegCp5(i)];
            lrow5=lrow5+1;
        end
    end
    usurface5=sortrows(usurf5,3);
    lsurface5=sortrows(lsurf5,3);
    Cn5=-
    trapz(usurface5(:,3),usurface5(:,5))+trapz(lsurface5(:,3),lsurface5(:,5)
    );
    Ct5=-
    trapz(lsurface5(:,4),lsurface5(:,5))+trapz(usurface5(:,4),usurface5(:,5)
    );
    Ct5=-1*Ct5;
    Cthrust5=(Cn5*cosd(twist_angle5))-(Ct5*sind(twist_angle5));
    Ctorque5=(Cn5*sind(twist_angle5))+(Ct5*cosd(twist_angle5));
    tolerance=1e-5;
    a=0;
    a_dash=0;
    a_new=0;
    diff_btwn_a_and_a_anew=300*tolerance;
    while diff_btwn_a_and_a_anew>tolerance
        phi5=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated5(1,:)));
        f_tip=0.5*Blades*( (R-
        z_translated5(1,:))/(z_translated5(1,:)*sin(phi5)));
        F_tip=(2/pi)*acos(exp(-1*f_tip));

```

```

        f_hub=0.5*Blades*((z_translated5(1,:)-
hubradius)/(z_translated5(1,:)*sin(phi5)));
        F_hub=(2/pi)*acos(exp(-1*f_hub));
        F=F_tip*F_hub;
        %Buhl Model
        if a<=0.4

a_new=1/(((8*pi*z_translated5(1,:)*F*sin(phi5)*sin(phi5))/(chord5*Blades
*Cthrust5))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-
4))))^0.5)/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated5(1,:)*F*sin(phi5)*cos(phi5))/(chord5*Blades
*Ctorque5))-1);
        end
        CL5=(Cthrust5*cos(phi5))+(Ctorque5*sin(phi5));
        CD5=(Cthrust5*sin(phi5))-(Ctorque5*cos(phi5));
        AoA5=(phi5*180/pi)-twist_angle5;
        elseif j==6
            import6=xlsread('PressureValues6.xls'); % Read excel file
            x6=import6(:,1);
            y6=import6(:,2);
            z6=import6(:,3);
            Pressure6=import6(:,4);
            twist_angle6=twist(j)+Bladepitchangle;
            % Distance between the leading edge and twist axis %
            twist_axis6=twistaxis(j);
            chord6=chord_length(j);
            x_rotated6=(x6*cosd(twist_angle6))+(y6*sind(twist_angle6));
            y_rotated6=(-x6*sind(twist_angle6))+(y6*cosd(twist_angle6));
            z_rotated6=z6;
            x_translated6=x_rotated6+twist_axis6;
            y_translated6=y_rotated6;
            z_translated6=z_rotated6;
            Cp6=(Pressure6-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated6(1,:)).^2));
            NegCp6=-1*Cp6(:,1);
            xbyc6=x_translated6/chord6;
            ybyc6=y_translated6/chord6;
            % Initialize row index for upper and lower surfaces
            urow6=1;
            lrow6=1;
            % Initialize upper and lower surface arrays
            usurf6=[];
            lsurf6=[];
            i=1;
            for i=1:size(import6,1)
                if y_translated6(i)>0.0000000
                    usurf6(urow6,:)=[x_translated6(i) y_translated6(i) xbyc6(i)
ybyc6(i) Cp6(i) NegCp6(i)];
                    urow6=urow6+1;
                else
                    lsurf6(lrow6,:)=[x_translated6(i) y_translated6(i) xbyc6(i)
ybyc6(i) Cp6(i) NegCp6(i)];
                    lrow6=lrow6+1;
                end
            end

```



```

end
usurface6=sortrows(usurf6,3);
lsurface6=sortrows(lsurf6,3);
Cn6=-
trapz(usurface6(:,3),usurface6(:,5))+trapz(lsurface6(:,3),lsurface6(:,5)
);
Ct6=-
trapz(lsurface6(:,4),lsurface6(:,5))+trapz(usurface6(:,4),usurface6(:,5)
);
Ct6=-1*Ct6;
Cthrust6=(Cn6*cosd(twist_angle6))-(Ct6*sind(twist_angle6));
Ctorque6=(Cn6*sind(twist_angle6))+(Ct6*cosd(twist_angle6));
tolerance=1e-3;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phi6=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated6(1,:)));
    f_tip=0.5*Blades*(R-
z_translated6(1,:))/(z_translated6(1,:)*sin(phi6));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated6(1,:)-
hubradius)/(z_translated6(1,:)*sin(phi6)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated6(1,:)*F*sin(phi6)*sin(phi6))/(chord6*Blades
*Cthrust6))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*((C_T*(50-(36*F)))+(12*F*((3*F)-
4))))^0.5)/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated6(1,:)*F*sin(phi6)*cos(phi6))/(chord6*Blade
s*Ctorque6))-1);
end
CL6=(Cthrust6*cos(phi6))+(Ctorque6*sin(phi6));
CD6=(Cthrust6*sin(phi6))-(Ctorque6*cos(phi6));
AoA6=(phi6*180/pi)-twist_angle6;
elseif j==7
    import7=xlsread('PressureValues7.xls'); % Read excel file
x7=import7(:,1);
y7=import7(:,2);
z7=import7(:,3);
Pressure7=import7(:,4);
twist_angle7=twist(j)+Bladepitchangle;
% Distance between the leading edge and twist axis %
twist_axis7=twistaxis(j);
chord7=chord_length(j);
x_rotated7=(x7*cosd(twist_angle7))+(y7*sind(twist_angle7));
y_rotated7=(-x7*sind(twist_angle7))+(y7*cosd(twist_angle7));
z_rotated7=z7;
x_translated7=x_rotated7+twist_axis7;
y_translated7=y_rotated7;
z_translated7=z_rotated7;

```

```

Cp7=(Pressure7-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated7(1,:)).^2));
NegCp7=-1*Cp7(:,1);
xbyc7=x_translated7/chord7;
ybyc7=y_translated7/chord7;
% Initialize row index for upper and lower surfaces
urow7=1;
lrow7=1;
% Initialize upper and lower surface arrays
usurf7=[];
lsurf7=[];
i=1;
for i=1:size(import7,1)
    if y_translated7(i)>0.0000000
        usurf7(urow7,:)= [x_translated7(i) y_translated7(i) xbyc7(i)
ybyc7(i) Cp7(i) NegCp7(i)];
        urow7=urow7+1;
    else
        lsurf7(lrow7,:)= [x_translated7(i) y_translated7(i) xbyc7(i)
ybyc7(i) Cp7(i) NegCp7(i)];
        lrow7=lrow7+1;
    end
end
usurface7=sortrows(usurf7,3);
lsurface7=sortrows(lsurf7,3);
subplot(3,2,2)
exp_47=xlsread('exp_47.xls');
exp_x_47=exp_47(:,1);
exp_Cp_47=exp_47(:,2);
lower_range=exp_47(:,5);
upper_range=exp_47(:,6);
width=3;
height=3;
axes_line_width=2;
font_size=8;
line_width=1.5;
marker_size=10;
pos=get(gcf, 'Position');
set(gcf, 'Position', [pos(1) pos(2) width*100 height*100]);
set(gca, 'FontSize', font_size, 'LineWidth', axes_line_width);
plot(xbyc7, NegCp7(:,1), 'blue --
', 'LineWidth', axes_line_width, 'MarkerSize', marker_size);
hold on
errorbar(exp_x_47, exp_Cp_47, lower_range, upper_range, 'red
*', 'LineWidth', axes_line_width, 'MarkerSize', marker_size);
xlim([-0.1, 1.1]);
ylim([-2, 6]);
xlabel('X/Chord');
ylabel('-Cp');
str='r/R';str='r/R';
prompt = 'r/R ';
str1 = input(prompt, 's');
title([str ': ' str1]);
Cn7=-
trapz(usurface7(:,3),usurface7(:,5))+trapz(lsurface7(:,3),lsurface7(:,5)
);
Ct7=-
trapz(lsurface7(:,4),lsurface7(:,5))+trapz(usurface7(:,4),usurface7(:,5)
);
Ct7=-1*Ct7;
Cthrust7=(Cn7*cosd(twist_angle7))-(Ct7*sind(twist_angle7));

```

```

Ctorque7=(Cn7*sind(twist_angle7))+(Ct7*cosd(twist_angle7));
tolerence=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerence
    phi7=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated7(1,:)));
    f_tip=0.5*Blades*(R-
z_translated7(1,:)/(z_translated7(1,:)*sin(phi7)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated7(1,:)-
hubradius)/(z_translated7(1,:)*sin(phi7)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated7(1,:)*F*sin(phi7)*sin(phi7))/(chord7*Blades
*Cthrust7))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*((C_T*(50-(36*F)))+(12*F*((3*F)-
4)))^0.5))/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated7(1,:)*F*sin(phi7)*cos(phi7))/(chord7*Blade
s*Ctorque7))-1);
    end
    CL7=(Cthrust7*cos(phi7))+(Ctorque7*sin(phi7));
    CD7=(Cthrust7*sin(phi7))-(Ctorque7*cos(phi7));
    AoA7=(phi7*180/pi)-twist_angle7;
elseif j==8
    import8=xlsread('PressureValues8.xls'); % Read excel file
    x8=import8(:,1);
    y8=import8(:,2);
    z8=import8(:,3);
    Pressure8=import8(:,4);
    twist_angle8=twist(j)+Bladepitchangle;
    % Distance between the leading edge and twist axis %
    twist_axis8=twistaxis(j);
    chord8=chord_length(j);
    x_rotated8=(x8*cosd(twist_angle8))+(y8*sind(twist_angle8));
    y_rotated8=(-x8*sind(twist_angle8))+(y8*cosd(twist_angle8));
    z_rotated8=z8;
    x_translated8=x_rotated8+twist_axis8;
    y_translated8=y_rotated8;
    z_translated8=z_rotated8;
    Cp8=(Pressure8-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated8(1,:)).^2));
    NegCp8=-1*Cp8(:,1);
    xbyc8=x_translated8/chord8;
    ybyc8=y_translated8/chord8;
    % Initialize row index for upper and lower surfaces
    urow8=1;
    lrow8=1;
    % Initialize upper and lower surface arrays
    usurf8=[];
    lsurf8=[];

```

```

i=1;
for i=1:size(import8,1)
    if y_translated8(i)>0.0000000
        usurf8(urow8,:)= [x_translated8(i) y_translated8(i) xbyc8(i)
ybyc8(i) Cp8(i) NegCp8(i)];
        urow8=urow8+1;
    else
        lsurf8(lrow8,:)= [x_translated8(i) y_translated8(i) xbyc8(i)
ybyc8(i) Cp8(i) NegCp8(i)];
        lrow8=lrow8+1;
    end
end
usurface8=sortrows(usurf8,3);
lsurface8=sortrows(lsurf8,3);
Cn8=-
trapz(usurface8(:,3),usurface8(:,5))+trapz(lsurface8(:,3),lsurface8(:,5)
);
Ct8=-
trapz(lsurface8(:,4),lsurface8(:,5))+trapz(usurface8(:,4),usurface8(:,5)
);
Ct8=-1*Ct8;
Cthrust8=(Cn8*cosd(twist_angle8)-(Ct8*sind(twist_angle8));
Ctorque8=(Cn8*sind(twist_angle8)+(Ct8*cosd(twist_angle8));
tolerence=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerence
    phi8=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated8(1,:)));
    f_tip=0.5*Blades*(R-
z_translated8(1,:)/(z_translated8(1,:)*sin(phi8)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated8(1,:)-
hubradius)/(z_translated8(1,:)*sin(phi8)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated8(1,:)*F*sin(phi8)*sin(phi8))/(chord8*Blades
*Cthrust8))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-
4)))^0.5)/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated8(1,:)*F*sin(phi8)*cos(phi8))/(chord8*Blade
s*Ctorque8))-1);
    end
    CL8=(Cthrust8*cos(phi8)+(Ctorque8*sin(phi8));
    CD8=(Cthrust8*sin(phi8)-(Ctorque8*cos(phi8));
    AoA8=(phi8*180/pi)-twist_angle8;
elseif j==9
    import9=xlsread('PressureValues9.xls'); % Read excel file
x9=import9(:,1);
y9=import9(:,2);
z9=import9(:,3);

```

```

Pressure9=import9(:,4);
twist_angle9=twist(j)+Bladepitchangle;
% Distance between the leading edge and twist axis %
twist_axis9=twistaxis(j);
chord9=chord_length(j);
x_rotated9=(x9*cosd(twist_angle9))+(y9*sind(twist_angle9));
y_rotated9=(-x9*sind(twist_angle9))+(y9*cosd(twist_angle9));
z_rotated9=z9;
x_translated9=x_rotated9+twist_axis9;
y_translated9=y_rotated9;
z_translated9=z_rotated9;
Cp9=(Pressure9-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated9(1,:)).^2));
NegCp9=-1*Cp9(:,1);
xbyc9=x_translated9/chord9;
ybyc9=y_translated9/chord9;
% Initialize row index for upper and lower surfaces
urow9=1;
lrow9=1;
% Initialize upper and lower surface arrays
usurf9=[];
lsurf9=[];
i=1;
for i=1:size(import9,1)
    if y_translated9(i)>0.0000000
        usurf9(urow9,:)=x_translated9(i) y_translated9(i) xbyc9(i)
ybyc9(i) Cp9(i) NegCp9(i)];
        urow9=urow9+1;
    else
        lsurf9(lrow9,:)=x_translated9(i) y_translated9(i) xbyc9(i)
ybyc9(i) Cp9(i) NegCp9(i)];
        lrow9=lrow9+1;
    end
end
usurface9=sortrows(usurf9,3);
lsurface9=sortrows(lsurf9,3);
Cn9=-
trapz(usurface9(:,3),usurface9(:,5))+trapz(lsurface9(:,3),lsurface9(:,5)
);
Ct9=-
trapz(lsurface9(:,4),lsurface9(:,5))+trapz(usurface9(:,4),usurface9(:,5)
);
Ct9=-1*Ct9;
Cthrust9=(Cn9*cosd(twist_angle9))-(Ct9*sind(twist_angle9));
Ctorque9=(Cn9*sind(twist_angle9))+(Ct9*cosd(twist_angle9));
tolerence=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerence
    phi9=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated9(1,:)));
    f_tip=0.5*Blades*((R-
z_translated9(1,:))/(z_translated9(1,:)*sin(phi9)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated9(1,:)-
hubradius)/(z_translated9(1,:)*sin(phi9)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

```

```

a_new=1/(((8*pi*z_translated9(1,:)*F*sin(phi9)*sin(phi9))/(chord9*Blades
*Cthrust9))+1);
    else
        C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
        a_new=((18*F)-20-(3*((C_T*(50-(36*F)))+(12*F*((3*F)-
4))))^0.5)/((36*F)-50);
    end
    diff_btwn_a_and_a_new=abs(a-a_new);
    a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated9(1,:)*F*sin(phi9)*cos(phi9))/(chord9*Blade
s*Ctorque9))-1);
end
CL9=(Cthrust9*cos(phi9))+(Ctorque9*sin(phi9));
CD9=(Cthrust9*sin(phi9))-(Ctorque9*cos(phi9));
AoA9=(phi9*180/pi)-twist_angle9;
elseif j==10
    import10=xlsread('PressureValues10.xls'); % Read excel file
x10=import10(:,1);
y10=import10(:,2);
z10=import10(:,3);
Pressure10=import10(:,4);
twist_angle10=twist(j)+Bladepitchangle;
% Distance between the leading edge and twist axis %
twist_axis10=twistaxis(j);
chord10=chord_length(j);
x_rotated10=(x10*cosd(twist_angle10))+(y10*sind(twist_angle10));
y_rotated10=(-x10*sind(twist_angle10))+(y10*cosd(twist_angle10));
z_rotated10=z10;
x_translated10=x_rotated10+twist_axis10;
y_translated10=y_rotated10;
z_translated10=z_rotated10;
Cp10=(Pressure10-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated10(1,:)).^2));
NegCp10=-1*Cp10(:,1);
xbyc10=x_translated10/chord10;
ybyc10=y_translated10/chord10;
% Initialize row index for upper and lower surfaces
urow10=1;
lrow10=1;
% Initialize upper and lower surface arrays
usurf10=[];
lsurf10=[];
i=1;
for i=1:size(import10,1)
    if y_translated10(i)>0.0000000
        usurf10(urow10,:)=[x_translated10(i) y_translated10(i) xbyc10(i)
ybyc10(i) Cp10(i) NegCp10(i)];
        urow10=urow10+1;
    else
        lsurf10(lrow10,:)=[x_translated10(i) y_translated10(i) xbyc10(i)
ybyc10(i) Cp10(i) NegCp10(i)];
        lrow10=lrow10+1;
    end
end
end
usurface10=sortrows(usurf10,3);
lsurface10=sortrows(lsurf10,3);
Cn10=-
trapz(usurface10(:,3),usurface10(:,5))+trapz(lsurface10(:,3),lsurface10(
(:,5)));

```

```

Ct10=-
trapz(lsurface10(:,4),lsurface10(:,5))+trapz(usurface10(:,4),usurface10(
(:,5)));
Ct10=-1*Ct10;
Cthrust10=(Cn10*cosd(twist_angle10)-(Ct10*sind(twist_angle10));
Ctorque10=(Cn10*sind(twist_angle10)+(Ct10*cosd(twist_angle10));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phi10=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated10(1,:)));
    f_tip=0.5*Blades*(R-
z_translated10(1,:)/(z_translated10(1,:)*sin(phi10)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated10(1,:)-
hubradius)/(z_translated10(1,:)*sin(phi10)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated10(1,:)*F*sin(phi10)*sin(phi10))/(chord10*B1
ades*Cthrust10))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-
4)))^0.5)/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated10(1,:)*F*sin(phi10)*cos(phi10))/(chord10*B
lades*Ctorque10))-1);
    end
    CL10=(Cthrust10*cos(phi10)+(Ctorque10*sin(phi10));
    CD10=(Cthrust10*sin(phi10)-(Ctorque10*cos(phi10));
    AoA10=(phi10*180/pi)-twist_angle10;
elseif j==11
    import11=xlsread('PressureValues11.xls'); % Read excel
    file
    x11=import11(:,1);
    y11=import11(:,2);
    z11=import11(:,3);
    Pressure11=import11(:,4);
    twist_angle11=twist(j)+Bladepitchangle;
    % Distance between the leading edge and twist axis %
    twist_axis11=twistaxis(j);
    chord11=chord_length(j);
    x_rotated11=(x11*cosd(twist_angle11)+(y11*sind(twist_angle11));
    y_rotated11=(-x11*sind(twist_angle11)+(y11*cosd(twist_angle11));
    z_rotated11=z11;
    x_translated11=x_rotated11+twist_axis11;
    y_translated11=y_rotated11;
    z_translated11=z_rotated11;
    Cp11=(Pressure11-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated11(1,:)).^2));
    NegCp11=-1*Cp11(:,1);
    xbyc11=x_translated11/chord11;
    ybyc11=y_translated11/chord11;

```

```

% Initialize row index for upper and lower surfaces
urow11=1;
lrow11=1;
% Initialize upper and lower surface arrays
usurf11=[];
lsurf11=[];
i=1;
for i=1:size(import11,1)
    if y_translated11(i)>0.0000000
        usurf11(urow11,:)= [x_translated11(i) y_translated11(i) xbyc11(i)
ybyc11(i) Cp11(i) NegCp11(i)];
        urow11=urow11+1;
    else
        lsurf11(lrow11,:)= [x_translated11(i) y_translated11(i) xbyc11(i)
ybyc11(i) Cp11(i) NegCp11(i)];
        lrow11=lrow11+1;
    end
end
end
usurface11=sortrows(usurf11,3);
lsurface11=sortrows(lsurf11,3);
subplot(3,2,3)
exp_63=xlsread('exp_63.xls');
exp_x_63=exp_63(:,1);
exp_Cp_63=exp_63(:,2);
lower_range=exp_63(:,5);
upper_range=exp_63(:,6);
width=3;
height=3;
axes_line_width=2;
font_size=8;
line_width=1.5;
marker_size=10;
pos = get(gcf, 'Position');
set(gcf, 'Position', [pos(1) pos(2) width*100 height*100]);
set(gca, 'FontSize', font_size, 'LineWidth', axes_line_width);
plot(xbyc11, NegCp11(:,1), 'blue --
', 'LineWidth', axes_line_width, 'MarkerSize', marker_size);
hold on
errorbar(exp_x_63, exp_Cp_63, lower_range, upper_range, 'red
*', 'LineWidth', axes_line_width, 'MarkerSize', marker_size);
xlim([-0.1, 1.1]);
ylim([-2, 6]);
xlabel('X/Chord');
ylabel('-Cp');
str='r/R';str='r/R';
prompt = 'r/R ';
str1 = input(prompt, 's');
title([str ': ' str1]);
Cn11=-
trapz(usurface11(:,3),usurface11(:,5))+trapz(lsurface11(:,3),lsurface11(
(:,5)));
Ct11=-
trapz(lsurface11(:,4),lsurface11(:,5))+trapz(usurface11(:,4),usurface11(
(:,5)));
Ct11=-1*Ct11;
Cthrust11=(Cn11*cosd(twist_angle11)-(Ct11*sind(twist_angle11));
Ctorque11=(Cn11*sind(twist_angle11)+(Ct11*cosd(twist_angle11));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;

```



```

diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phil1=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated11(1,:)));
    f_tip=0.5*Blades*(R-
z_translated11(1,:)/(z_translated11(1,:)*sin(phil1)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated11(1,:)-
hubradius)/(z_translated11(1,:)*sin(phil1)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated11(1,:)*F*sin(phil1)*sin(phil1))/(chord11*Blades*Cthrust11))+1);
    else
        C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
        a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-4)))^0.5))/((36*F)-50);
    end
    diff_btwn_a_and_a_anew=abs(a-a_new);
    a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated11(1,:)*F*sin(phil1)*cos(phil1))/(chord11*Blades*Ctorque11))-1);
end
CL11=(Cthrust11*cos(phil1))+(Ctorque11*sin(phil1));
CD11=(Cthrust11*sin(phil1))-(Ctorque11*cos(phil1));
AoA11=(phil1*180/pi)-twist_angle11;
elseif j==12
import12=xlsread('PressureValues12.xls'); % Read excel file
x12=import12(:,1);
y12=import12(:,2);
z12=import12(:,3);
Pressure12=import12(:,4);
twist_angle12=twist(j)+Bladepitchangle;
% Distance between the leading edge and twist axis %
twist_axis12=twistaxis(j);
chord12=chord_length(j);
x_rotated12=(x12*cosd(twist_angle12))+(y12*sind(twist_angle12));
y_rotated12=(-x12*sind(twist_angle12))+(y12*cosd(twist_angle12));
z_rotated12=z12;
x_translated12=x_rotated12+twist_axis12;
y_translated12=y_rotated12;
z_translated12=z_rotated12;
Cp12=(Pressure12-Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated12(1,:)).^2));
NegCp12=-1*Cp12(:,1);
xbyc12=x_translated12/chord12;
ybyc12=y_translated12/chord12;
% Initialize row index for upper and lower surfaces
urow12=1;
lrow12=1;
% Initialize upper and lower surface arrays
usurf12=[];
lsurf12=[];
i=1;
for i=1:size(import12,1)
    if y_translated12(i)>0.0000000
        usurf12(urow12,:)=[x_translated12(i) y_translated12(i) xbyc12(i) ybyc12(i) Cp12(i) NegCp12(i)];
        urow12=urow12+1;
    else
        lsurf12(lrow12,:)=[x_translated12(i) y_translated12(i) xbyc12(i) ybyc12(i) Cp12(i) NegCp12(i)];
        lrow12=lrow12+1;
    end
end

```

```

        urow12=urow12+1;
    else
        lsurf12(lrow12,:)= [x_translated12(i) y_translated12(i) xbyc12(i)
ybyc12(i) Cp12(i) NegCp12(i)];
        lrow12=lrow12+1;
    end
end
usurface12=sortrows(usurf12,3);
lsurface12=sortrows(lsurf12,3);
Cn12=-
trapz(usurface12(:,3),usurface12(:,5))+trapz(lsurface12(:,3),lsurface12(
(:,5)));
Ct12=-
trapz(lsurface12(:,4),lsurface12(:,5))+trapz(usurface12(:,4),usurface12(
(:,5)));
Ct12=-1*Ct12;
Cthrust12=(Cn12*cosd(twist_angle12))-(Ct12*sind(twist_angle12));
Ctorque12=(Cn12*sind(twist_angle12))+(Ct12*cosd(twist_angle12));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phil2=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated12(1,:)));
    f_tip=0.5*Blades*(R-
z_translated12(1,:)/(z_translated12(1,:)*sin(phil2)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated12(1,:)-
hubradius)/(z_translated12(1,:)*sin(phil2)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated12(1,:)*F*sin(phil2)*sin(phil2))/(chord12*Bl
ades*Cthrust12))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-
4)))^0.5))/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated12(1,:)*F*sin(phil2)*cos(phil2))/(chord12*Bl
ades*Ctorque12))-1);
    end
    CL12=(Cthrust12*cos(phil2))+(Ctorque12*sin(phil2));
    CD12=(Cthrust12*sin(phil2))-(Ctorque12*cos(phil2));
    AoA12=(phil2*180/pi)-twist_angle12;
elseif j==13
    importl3=xlsread('PressureValues13.xls'); % Read excel file
    x13=importl3(:,1);
    y13=importl3(:,2);
    z13=importl3(:,3);
    Pressure13=importl3(:,4);
    twist_angle13=twist(j)+Bladepitchangle;
    % Distance between the leading edge and twist axis %
    twist_axis13=twistaxis(j);
    chord13=chord_length(j);

```

```

x_rotated13=(x13*cosd(twist_angle13))+(y13*sind(twist_angle13));
y_rotated13=(-x13*sind(twist_angle13))+(y13*cosd(twist_angle13));
z_rotated13=z13;
x_translated13=x_rotated13+twist_axis13;
y_translated13=y_rotated13;
z_translated13=z_rotated13;
Cp13=(Pressure13-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated13(1,:)).^2));
NegCp13=-1*Cp13(:,1);
xbyc13=x_translated13/chord13;
ybyc13=y_translated13/chord13;
% Initialize row index for upper and lower surfaces
urow13=1;
lrow13=1;
% Initialize upper and lower surface arrays
usurf13=[];
lsurf13=[];
i=1;
for i=1:size(import13,1)
    if y_translated13(i)>0.0000000
        usurf13(urow13,:)=[x_translated13(i) y_translated13(i) xbyc13(i)
ybyc13(i) Cp13(i) NegCp13(i)];
        urow13=urow13+1;
    else
        lsurf13(lrow13,:)=[x_translated13(i) y_translated13(i) xbyc13(i)
ybyc13(i) Cp13(i) NegCp13(i)];
        lrow13=lrow13+1;
    end
end
usurface13=sortrows(usurf13,3);
lsurface13=sortrows(lsurf13,3);
Cn13=-
trapz(usurface13(:,3),usurface13(:,5))+trapz(lsurface13(:,3),lsurface13(
(:,5)));
Ct13=-
trapz(lsurface13(:,4),lsurface13(:,5))+trapz(usurface13(:,4),usurface13(
(:,5)));
Ct13=-1*Ct13;
Cthrust13=(Cn13*cosd(twist_angle13))-(Ct13*sind(twist_angle13));
Ctorque13=(Cn13*sind(twist_angle13))+(Ct13*cosd(twist_angle13));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phi13=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated13(1,:)));
    f_tip=0.5*Blades*((R-
z_translated13(1,:))/(z_translated13(1,:)*sin(phi13)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated13(1,:)-
hubradius)/(z_translated13(1,:)*sin(phi13)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated13(1,:)*F*sin(phi13)*sin(phi13))/(chord13*Bl
ades*Cthrust13))+1);
    else
        C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));

```

```

a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*(3*F)-
4)))^0.5))/((36*F)-50);
end
diff_btwn_a_and_a_anew=abs(a-a_new);
a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated13(1,:)*F*sin(phi13)*cos(phi13))/(chord13*B
lades*Ctorque13))-1);
end
CL13=(Cthrust13*cos(phi13))+(Ctorque13*sin(phi13));
CD13=(Cthrust13*sin(phi13))-(Ctorque13*cos(phi13));
AoA13=(phi13*180/pi)-twist_angle13;
elseif j==14
importl4=xlsread('PressureValues14.xls'); % Read excel file
x14=importl4(:,1);
y14=importl4(:,2);
z14=importl4(:,3);
Pressure14=importl4(:,4);
twist_angle14=twist(j)+Bladepitchangle;
% Distance between the leading edge and twist axis %
twist_axis14=twistaxis(j);
chord14=chord_length(j);
x_rotated14=(x14*cosd(twist_angle14))+(y14*sind(twist_angle14));
y_rotated14=(-x14*sind(twist_angle14))+(y14*cosd(twist_angle14));
z_rotated14=z14;
x_translated14=x_rotated14+twist_axis14;
y_translated14=y_rotated14;
z_translated14=z_rotated14;
Cp14=(Pressure14-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated14(1,:)).^2));
NegCp14=-1*Cp14(:,1);
xbyc14=x_translated14/chord14;
ybyc14=y_translated14/chord14;
% Initialize row index for upper and lower surfaces
urow14=1;
lrow14=1;
% Initialize upper and lower surface arrays
usurf14=[];
lsurf14=[];
i=1;
for i=1:size(importl4,1)
if y_translated14(i)>0.0000000
usurf14(urow14,:)=[x_translated14(i) y_translated14(i) xbyc14(i)
ybyc14(i) Cp14(i) NegCp14(i)];
urow14=urow14+1;
else
lsurf14(lrow14,:)=[x_translated14(i) y_translated14(i) xbyc14(i)
ybyc14(i) Cp14(i) NegCp14(i)];
lrow14=lrow14+1;
end
end
end
usurface14=sortrows(usurf14,3);
lsurface14=sortrows(lsurf14,3);
h1=subplot(3,2,4);
exp_80=xlsread('exp_80.xls');
exp_x_80=exp_80(:,1);
exp_Cp_80=exp_80(:,2);
lower_range=exp_80(:,5);
upper_range=exp_80(:,6);
width=3;
height=3;

```

```

axes_line_width=2;
font_size=8;
line_width=1.5;
marker_size=10;
pos = get(gcf, 'Position');
set(gcf, 'Position', [pos(1) pos(2) width*100 height*100]);
set(gca, 'FontSize', font_size, 'LineWidth', axes_line_width);
plot(xbyc14, NegCp14(:,1), 'blue --', 'LineWidth', axes_line_width, 'MarkerSize', marker_size);
hold on
errorbar(exp_x_80, exp_Cp_80, lower_range, upper_range, 'red', 'LineWidth', axes_line_width, 'MarkerSize', marker_size);
xlim([-0.1, 1.1]);
ylim([-2, 6]);
xlabel('X/Chord');
ylabel('-Cp');
str='r/R'; str='r/R';
prompt = 'r/R ';
str1 = input(prompt, 's');
title([str ': ' str1]);
Cn14=-
trapz(usurface14(:,3), usurface14(:,5))+trapz(lsurface14(:,3), lsurface14(:,5));
Ct14=-
trapz(lsurface14(:,4), lsurface14(:,5))+trapz(usurface14(:,4), usurface14(:,5));
Ct14=-1*Ct14;
Cthrust14=(Cn14*cosd(twist_angle14))-(Ct14*sind(twist_angle14));
Ctorque14=(Cn14*sind(twist_angle14))+(Ct14*cosd(twist_angle14));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phi14=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated14(1,:)));
    f_tip=0.5*Blades*((R-
z_translated14(1,:))/(z_translated14(1,:)*sin(phi14)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated14(1,:)-
hubradius)/(z_translated14(1,:)*sin(phi14)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated14(1,:)*F*sin(phi14)*sin(phi14))/(chord14*Blades*Cthrust14))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-4)))^0.5))/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated14(1,:)*F*sin(phi14)*cos(phi14))/(chord14*Blades*Ctorque14))-1);
    end
    CL14=(Cthrust14*cos(phi14))+(Ctorque14*sin(phi14));
    CD14=(Cthrust14*sin(phi14))-(Ctorque14*cos(phi14));

```

```

AoA14=(phi14*180/pi)-twist_angle14;
elseif j==15
    import15=xlsread('PressureValues15.xls'); % Read excel file
x15=import15(:,1);
y15=import15(:,2);
z15=import15(:,3);
Pressure15=import15(:,4);
twist_angle15=twist(j)+Bladepitchangle;
% Distance between the leading edge and twist axis %
twist_axis15=twistaxis(j);
chord15=chord_length(j);
x_rotated15=(x15*cosd(twist_angle15))+(y15*sind(twist_angle15));
y_rotated15=(-x15*sind(twist_angle15))+(y15*cosd(twist_angle15));
z_rotated15=z15;
x_translated15=x_rotated15+twist_axis15;
y_translated15=y_rotated15;
z_translated15=z_rotated15;
Cp15=(Pressure15-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated15(1,:)).^2));
NegCp15=-1*Cp15(:,1);
xbyc15=x_translated15/chord15;
ybyc15=y_translated15/chord15;
% Initialize row index for upper and lower surfaces
urow15=1;
lrow15=1;
% Initialize upper and lower surface arrays
usurf15=[];
lsurf15=[];
i=1;
for i=1:size(import15,1)
    if y_translated15(i)>0.0000000
        usurf15(urow15,:)=[x_translated15(i) y_translated15(i) xbyc15(i)
ybyc15(i) Cp15(i) NegCp15(i)];
        urow15=urow15+1;
    else
        lsurf15(lrow15,:)=[x_translated15(i) y_translated15(i) xbyc15(i)
ybyc15(i) Cp15(i) NegCp15(i)];
        lrow15=lrow15+1;
    end
end
usurface15=sortrows(usurf15,3);
lsurface15=sortrows(lsurf15,3);
Cn15=-
trapz(usurface15(:,3),usurface15(:,5))+trapz(lsurface15(:,3),lsurface15(
(:,5)));
Ct15=-
trapz(lsurface15(:,4),lsurface15(:,5))+trapz(usurface15(:,4),usurface15(
(:,5)));
Ct15=-1*Ct15;
Cthrust15=(Cn15*cosd(twist_angle15))-(Ct15*sind(twist_angle15));
Ctorque15=(Cn15*sind(twist_angle15))+(Ct15*cosd(twist_angle15));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phi15=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated15(1,:)));
    f_tip=0.5*Blades*(R-
z_translated15(1,:)/(z_translated15(1,:)*sin(phi15)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));

```

```

        f_hub=0.5*Blades*((z_translated15(1,:)-
hubradius)/(z_translated15(1,:)*sin(phi15)));
        F_hub=(2/pi)*acos(exp(-1*f_hub));
        F=F_tip*F_hub;
        %Buhl Model
        if a<=0.4

a_new=1/(((8*pi*z_translated15(1,:)*F*sin(phi15)*sin(phi15))/(chord15*Bl
ades*Cthrust15))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*(C_T*(50-(36*F)))+(12*F*((3*F)-
4))))^0.5)/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated15(1,:)*F*sin(phi15)*cos(phi15))/(chord15*B
lades*Ctorque15))-1);
        end
        CL15=(Cthrust15*cos(phi15))+(Ctorque15*sin(phi15));
        CD15=(Cthrust15*sin(phi15))-(Ctorque15*cos(phi15));
        AoA15=(phi15*180/pi)-twist_angle15;
        elseif j==16
            importl6=xlsread('PressureValues16.xls'); % Read excel file
            x16=importl6(:,1);
            y16=importl6(:,2);
            z16=importl6(:,3);
            Pressure16=importl6(:,4);
            twist_angle16=twist(j)+Bladepitchangle;
            % Distance between the leading edge and twist axis %
            twist_axis16=twistaxis(j);
            chord16=chord_length(j);
            x_rotated16=(x16*cosd(twist_angle16))+(y16*sind(twist_angle16));
            y_rotated16=(-x16*sind(twist_angle16))+(y16*cosd(twist_angle16));
            z_rotated16=z16;
            x_translated16=x_rotated16+twist_axis16;
            y_translated16=y_rotated16;
            z_translated16=z_rotated16;
            Cp16=(Pressure16-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated16(1,:)).^2));
            NegCp16=-1*Cp16(:,1);
            xbyc16=x_translated16/chord16;
            ybyc16=y_translated16/chord16;
            % Initialize row index for upper and lower surfaces
            urow16=1;
            lrow16=1;
            % Initialize upper and lower surface arrays
            usurf16=[];
            lsurf16=[];
            i=1;
            for i=1:size(importl6,1)
                if y_translated16(i)>0.0000000
                    usurf16(urow16,:)=[x_translated16(i) y_translated16(i) xbyc16(i)
ybyc16(i) Cp16(i) NegCp16(i)];
                    urow16=urow16+1;
                else
                    lsurf16(lrow16,:)=[x_translated16(i) y_translated16(i) xbyc16(i)
ybyc16(i) Cp16(i) NegCp16(i)];
                    lrow16=lrow16+1;
                end
            end

```

```

end
usurfacel6=sortrows(usurfl6,3);
lsurfacel6=sortrows(lsurfl6,3);
Cn16=-
trapz(usurfacel6(:,3),usurfacel6(:,5))+trapz(lsurfacel6(:,3),lsurfacel6(
(:,5)));
Ct16=-
trapz(lsurfacel6(:,4),lsurfacel6(:,5))+trapz(usurfacel6(:,4),usurfacel6(
(:,5)));
Ct16=-1*Ct16;
Cthrust16=(Cn16*cosd(twist_angle16))-(Ct16*sind(twist_angle16));
Ctorque16=(Cn16*sind(twist_angle16))+(Ct16*cosd(twist_angle16));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phil6=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated16(1,:)));
    f_tip=0.5*Blades*((R-
z_translated16(1,:))/(z_translated16(1,:)*sin(phil6)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated16(1,:)-
hubradius)/(z_translated16(1,:)*sin(phil6)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated16(1,:)*F*sin(phil6)*sin(phil6))/(chord16*B
lades*Cthrust16))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*((C_T*(50-(36*F)))+(12*F*((3*F)-
4)))^0.5))/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated16(1,:)*F*sin(phil6)*cos(phil6))/(chord16*B
lades*Ctorque16))-1);
end
CL16=(Cthrust16*cos(phil6))+(Ctorque16*sin(phil6));
CD16=(Cthrust16*sin(phil6))-(Ctorque16*cos(phil6));
AoA16=(phil6*180/pi)-twist_angle16;
elseif j==17
    importl7=xlsread('PressureValues17.xls'); % Read excel file
    x17=importl7(:,1);
    y17=importl7(:,2);
    z17=importl7(:,3);
    Pressure17=importl7(:,4);
    twist_angle17=twist(j)+Bladepitchangle;
    % Distance between the leading edge and twist axis %
    twist_axis17=twistaxis(j);
    chord17=chord_length(j);
    x_rotated17=(x17*cosd(twist_angle17))+(y17*sind(twist_angle17));
    y_rotated17=(-x17*sind(twist_angle17))+(y17*cosd(twist_angle17));
    z_rotated17=z17;
    x_translated17=x_rotated17+twist_axis17;
    y_translated17=y_rotated17;
    z_translated17=z_rotated17;

```



```

Cp17=(Pressure17-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated17(1,:)).^2));
NegCp17=-1*Cp17(:,1);
xbyc17=x_translated17/chord17;
ybyc17=y_translated17/chord17;
% Initialize row index for upper and lower surfaces
urow17=1;
lrow17=1;
% Initialize upper and lower surface arrays
usurf17=[];
lsurf17=[];
i=1;
for i=1:size(import17,1)
    if y_translated17(i)>0.0000000
        usurf17(urow17,:)=[x_translated17(i) y_translated17(i) xbyc17(i)
ybyc17(i) Cp17(i) NegCp17(i)];
        urow17=urow17+1;
    else
        lsurf17(lrow17,:)=[x_translated17(i) y_translated17(i) xbyc17(i)
ybyc17(i) Cp17(i) NegCp17(i)];
        lrow17=lrow17+1;
    end
end
usurface17=sortrows(usurf17,3);
lsurface17=sortrows(lsurf17,3);
Cn17=-
trapz(usurface17(:,3),usurface17(:,5))+trapz(lsurface17(:,3),lsurface17(
(:,5)));
Ct17=-
trapz(lsurface17(:,4),lsurface17(:,5))+trapz(usurface17(:,4),usurface17(
(:,5)));
Ct17=-1*Ct17;
Cthrust17=(Cn17*cosd(twist_angle17))-(Ct17*sind(twist_angle17));
Ctorque17=(Cn17*sind(twist_angle17))+(Ct17*cosd(twist_angle17));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phil7=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated17(1,:)));
    f_tip=0.5*Blades*((R-
z_translated17(1,:))/(z_translated17(1,:)*sin(phil7)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated17(1,:)-
hubradius)/(z_translated17(1,:)*sin(phil7)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<0.4

a_new=1/(((8*pi*z_translated17(1,:)*F*sin(phil7)*sin(phil7))/(chord17*Bla
ades*Cthrust17))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*((C_T*(50-(36*F)))+(12*F*((3*F)-
4)))^0.5))/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);
    end
end

```

```

a_dash=1/(((8*pi*z_translated17(1,:)*F*sin(phi17)*cos(phi17))/(chord17*Blades*Ctorque17))-1);
end
CL17=(Cthrust17*cos(phi17))+(Ctorque17*sin(phi17));
CD17=(Cthrust17*sin(phi17))-(Ctorque17*cos(phi17));
AoA17=(phi17*180/pi)-twist_angle17;
elseif j==18
    importl8=xlsread('PressureValues18.xls'); % Read excel
file
x18=importl8(:,1);
y18=importl8(:,2);
z18=importl8(:,3);
Pressure18=importl8(:,4);
twist_angle18=twist(j)+Bladepitchangle;
% Distance between the leading edge and twist axis %
twist_axis18=twistaxis(j);
chordl8=chord_length(j);
x_rotated18=(x18*cosd(twist_angle18))+(y18*sind(twist_angle18));
y_rotated18=(-x18*sind(twist_angle18))+(y18*cosd(twist_angle18));
z_rotated18=z18;
x_translated18=x_rotated18+twist_axis18;
y_translated18=y_rotated18;
z_translated18=z_rotated18;
Cp18=(Pressure18-Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated18(1,:)).^2));
NegCp18=-1*Cp18(:,1);
xbyc18=x_translated18/chordl8;
ybyc18=y_translated18/chordl8;
% Initialize row index for upper and lower surfaces
urow18=1;
lrow18=1;
% Initialize upper and lower surface arrays
usurf18=[];
lsurf18=[];
i=1;
for i=1:size(importl8,1)
    if y_translated18(i)>0.0000000
        usurf18(urow18,:)=[x_translated18(i) y_translated18(i) xbyc18(i)
ybyc18(i) Cp18(i) NegCp18(i)];
        urow18=urow18+1;
    else
        lsurf18(lrow18,:)=[x_translated18(i) y_translated18(i) xbyc18(i)
ybyc18(i) Cp18(i) NegCp18(i)];
        lrow18=lrow18+1;
    end
end
usurface18=sortrows(usurf18,3);
lsurface18=sortrows(lsurf18,3);
h2=subplot(3,2,5.5);
exp_95=xlsread('exp_95.xls');
exp_x_95=exp_95(:,1);
exp_Cp_95=exp_95(:,2);
lower_range=exp_95(:,5);
upper_range=exp_95(:,6);
width=3;
height=3;
axes_line_width=2;
font_size=8;
line_width=1.5;
marker_size=10;

```

```

pos=get(gcf, 'Position');
set(gcf, 'Position', [pos(1) pos(2) width*100 height*100]);
set(gca, 'FontSize', font_size, 'LineWidth', axes_line_width);
plot(xbysl8, NegCp18(:,1), 'blue --', 'LineWidth', axes_line_width, 'MarkerSize', marker_size);
hold on
errorbar(exp_x_95, exp_Cp_95, lower_range, upper_range, 'red *', 'LineWidth', axes_line_width, 'MarkerSize', marker_size);
xlim([-0.1, 1.1]);
ylim([-2, 4]);
lgd=legend('CFD', 'Experiment');
p1=get(h1, 'position');
p2=get(h2, 'position');
set(lgd, 'position', [p1(1) p2(2) p1(3) p1(4)]);
set(legend, 'color', 'none');
xlabel('X/Chord');
ylabel('-Cp');
str='r/R';str='r/R';
prompt = 'r/R ';
str1 = input(prompt, 's');
title([str ': ' str1]);
print('Cpplot', '-dpng', '-r300');
print('Cpplot1', '-depsc2', '-r300');
if ispc % Use Windows ghostscript call
    system('gswin64c -o -q -sDEVICE=png256 -dEPSCrop -r300 -Cpplot1.png Cpplot1.eps');
else % Use Unix/OSX ghostscript call
    system('gs -o -q -sDEVICE=png256 -dEPSCrop -r300 -oCpplot1_eps.png Cpplot1.eps');
end

Cn18=-
trapz(usurface18(:,3), usurface18(:,5))+trapz(lsurface18(:,3), lsurface18(:,5));
Ct18=-
trapz(lsurface18(:,4), lsurface18(:,5))+trapz(usurface18(:,4), usurface18(:,5));
Ct18=-1*Ct18;
Cthrust18=(Cn18*cosd(twist_angle18))-(Ct18*sind(twist_angle18));
Ctorque18=(Cn18*sind(twist_angle18))+(Ct18*cosd(twist_angle18));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phil8=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated18(1,:)));
    f_tip=0.5*Blades*(R-z_translated18(1,:)/(z_translated18(1,:)*sin(phil8)));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated18(1,:)-hubradius)/(z_translated18(1,:)*sin(phil8)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4
        a_new=1/(((8*pi*z_translated18(1,:)*F*sin(phil8)*sin(phil8))/(chord18*Blades*Cthrust18))+1);
    else
        C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
    end
end

```

```

a_new=((18*F)-20-(3*((C_T*(50-(36*F)))+(12*F*((3*F)-
4))))^0.5))/((36*F)-50);
end
diff_btwn_a_and_a_anew=abs(a-a_new);
a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated18(1,:)*F*sin(phi18)*cos(phi18))/(chord18*B
lades*Ctorque18))-1);
end
CL18=(Cthrust18*cos(phi18))+(Ctorque18*sin(phi18));
CD18=(Cthrust18*sin(phi18))-(Ctorque18*cos(phi18));
AoA18=(phi18*180/pi)-twist_angle18;
elseif j==19
import19=xlsread('PressureValues19.xls'); % Read
excel file
x19=import19(:,1);
y19=import19(:,2);
z19=import19(:,3);
Pressure19=import19(:,4);
twist_angle19=twist(j)+Bladepitchangle;
% Distance between the leading edge and twist axis %
twist_axis19=twistaxis(j);
chord19=chord_length(j);
x_rotated19=(x19*cosd(twist_angle19))+(y19*sind(twist_angle19));
y_rotated19=(-x19*sind(twist_angle19))+(y19*cosd(twist_angle19));
z_rotated19=z19;
x_translated19=x_rotated19+twist_axis19;
y_translated19=y_rotated19;
z_translated19=z_rotated19;
Cp19=(Pressure19-
Pr_atm)/(0.5*density*(U_wind.^2+(omega*z_translated19(1,:)).^2));
NegCp19=-1*Cp19(:,1);
xbyc19=x_translated19/chord19;
ybyc19=y_translated19/chord19;
% Initialize row index for upper and lower surfaces
urow19=1;
lrow19=1;
% Initialize upper and lower surface arrays
usurf19=[];
lsurf19=[];
i=1;
for i=1:size(import19,1)
if y_translated19(i)>0.0000000
usurf19(urow19,:)=[x_translated19(i) y_translated19(i) xbyc19(i)
ybyc19(i) Cp19(i) NegCp19(i)];
urow19=urow19+1;
else
lsurf19(lrow19,:)=[x_translated19(i) y_translated19(i) xbyc19(i)
ybyc19(i) Cp19(i) NegCp19(i)];
lrow19=lrow19+1;
end
end
usurface19=sortrows(usurf19,3);
lsurface19=sortrows(lsurf19,3);
Cn19=-
trapz(usurface19(:,3),usurface19(:,5))+trapz(lsurface19(:,3),lsurface19(
(:,5)));
Ct19=-
trapz(lsurface19(:,4),lsurface19(:,5))+trapz(usurface19(:,4),usurface19(
(:,5)));
Ct19=-1*Ct19;

```

```

Cthrust19=(Cn19*cosd(twist_angle19))-(Ct19*sind(twist_angle19));
Ctorque19=(Cn19*sind(twist_angle19))+(Ct19*cosd(twist_angle19));
tolerance=1e-5;
a=0;
a_dash=0;
a_new=0;
diff_btwn_a_and_a_anew=300*tolerance;
while diff_btwn_a_and_a_anew>tolerance
    phil9=atan(((1-a)*U_wind)/((1+a_dash)*omega*z_translated19(1,:)));
    f_tip=0.5*Blades*(R-
z_translated19(1,:))/(z_translated19(1,:)*sin(phil9));
    F_tip=(2/pi)*acos(exp(-1*f_tip));
    f_hub=0.5*Blades*((z_translated19(1,:)-
hubradius)/(z_translated19(1,:)*sin(phil9)));
    F_hub=(2/pi)*acos(exp(-1*f_hub));
    F=F_tip*F_hub;
    %Buhl Model
    if a<=0.4

a_new=1/(((8*pi*z_translated19(1,:)*F*sin(phil9)*sin(phil9))/(chord19*Bl
ades*Cthrust19))+1);
        else
            C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
            a_new=((18*F)-20-(3*((C_T*(50-(36*F)))+(12*F*((3*F)-
4))))^0.5)/((36*F)-50);
        end
        diff_btwn_a_and_a_anew=abs(a-a_new);
        a=0.5*(a+a_new);

a_dash=1/(((8*pi*z_translated19(1,:)*F*sin(phil9)*cos(phil9))/(chord19*Bl
ades*Ctorque19))-1);
    end
    CL19=(Cthrust19*cos(phil9))+(Ctorque19*sin(phil9));
    CD19=(Cthrust19*sin(phil9))-(Ctorque19*cos(phil9));
    AoA19=(phil9*180/pi)-twist_angle19;
end
end
fprintf('      radius      Cn      Ct      Cthrust      Ctorque
Phi      AoA      CL      CD \n');
[z_translated1(1,:) Cn1 Ct1 Cthrust1 Ctorque1 phil1 AoA1 CL1 CD1
z_translated2(1,:) Cn2 Ct2 Cthrust2 Ctorque2 phil2 AoA2 CL2 CD2
z_translated3(1,:) Cn3 Ct3 Cthrust3 Ctorque3 phil3 AoA3 CL3 CD3
z_translated4(1,:) Cn4 Ct4 Cthrust4 Ctorque4 phil4 AoA4 CL4 CD4
z_translated5(1,:) Cn5 Ct5 Cthrust5 Ctorque5 phil5 AoA5 CL5 CD5
z_translated6(1,:) Cn6 Ct6 Cthrust6 Ctorque6 phil6 AoA6 CL6 CD6
z_translated7(1,:) Cn7 Ct7 Cthrust7 Ctorque7 phil7 AoA7 CL7 CD7
z_translated8(1,:) Cn8 Ct8 Cthrust8 Ctorque8 phil8 AoA8 CL8 CD8
z_translated9(1,:) Cn9 Ct9 Cthrust9 Ctorque9 phil9 AoA9 CL9 CD9
z_translated10(1,:) Cn10 Ct10 Cthrust10 Ctorque10 phil10 AoA10
CL10 CD10
z_translated11(1,:) Cn11 Ct11 Cthrust11 Ctorque11 phil11 AoA11
CL11 CD11
z_translated12(1,:) Cn12 Ct12 Cthrust12 Ctorque12 phil12 AoA12
CL12 CD12
z_translated13(1,:) Cn13 Ct13 Cthrust13 Ctorque13 phil13 AoA13
CL13 CD13
z_translated14(1,:) Cn14 Ct14 Cthrust14 Ctorque14 phil14 AoA14
CL14 CD14
z_translated15(1,:) Cn15 Ct15 Cthrust15 Ctorque15 phil15 AoA15
CL15 CD15

```

```

        z_translated16(1,:) Cn16 Ct16 Cthrust16 Ctorque16 phi16 AoA16
CL16 CD16
        z_translated17(1,:) Cn17 Ct17 Cthrust17 Ctorque17 phi17 AoA17
CL17 CD17
        z_translated18(1,:) Cn18 Ct18 Cthrust18 Ctorque18 phi18 AoA18
CL18 CD18
        z_translated19(1,:) Cn19 Ct19 Cthrust19 Ctorque19 phi19 AoA19
CL19 CD19]

```

APPENDIX C: Proposed BEM MATLAB code

```

Blades=input('Number of blades:');
Bladepitchangle=input('Bladepitchangle:');
NofD=input('Number of datas:');
R=input('Radius of tip in metre:');
hubradius=input('Radius of root in metre:');
N=input('Enter number of blade elements in BEM:');
load NRELPHASEVIbladegeometrydata_new.dat;
given_R=NRELPHASEVIbladegeometrydata_new(:,1);
given_C=NRELPHASEVIbladegeometrydata_new(:,2);
given_twist=NRELPHASEVIbladegeometrydata_new(:,3);
for i=1:NofD
    radius(i)=given_R(i);
end
for i=1:NofD
    chord_length(i)=given_C(i);
end
for i=1:NofD
    twist(i)=given_twist(i);
end
U_wind = input('Enter the wind speed in m/s: ');
omega=7.5398223599963545;
density=1.225;
dr=(R-given_R(1))/N;
for i=1:N
    if (i == 1)
        radius_elem_midpoint(1)=given_R(1)+dr/2;
    else
        radius_elem_midpoint(i)=radius_elem_midpoint(i-1)+dr;
    end

    chord_elem_midpoint(i)=interp1(radius,chord_length,radius_elem_midpoint(
i),'spline');

    twist_elem_midpoint(i)=interp1(radius,twist,radius_elem_midpoint(i),'spl
ine');
end
viscosity=1.78*10^-5;
density=1.225;
omega=7.5398223599963545;
lamda=(omega*R)/U_wind;
TotalThrust = 0.0;
TotalTorque = 0.0;
fprintf('\n')
fprintf('radius      chord      localtwist      phi      AoA
a      Uwind*(1-a)      Urel      CL      CD      Cthrust
Ctorque      Dthrust      Dtorque      Pressuredrop      Re \n')
TotalThrust=0.0;
TotalTorque=0.0;
Dthrust=0;
Dtorque=0;
for i=1:N
    tolerance=1e-3;
    a=0;
    a_dash=0;
    a_new=0;
    lamda_r=lamda*radius_elem_midpoint(i)/R;
    diff_btwn_a_and_a_anew=300*tolerance;
    while diff_btwn_a_and_a_anew>tolerance
        %Calculation of Inflow Angle%

```

```

    phi=atan(((1-a)*U_wind)/((1+a_dash)*omega*radius_elem_midpoint(i)));
    %Calculation of Angle of Attack%
    AoA=(phi*180/pi)-twist_elem_midpoint(i)-Bladepitchangle;
    %Estimation of Cl and Cd %
    if ((radius_elem_midpoint(i)/R)<0.30)
    load section1_AoA_Cl_Cd.dat;
    AoA_list=section1_AoA_Cl_Cd(:,1);
    CL_list=section1_AoA_Cl_Cd(:,2);
    CD_list=section1_AoA_Cl_Cd(:,3);
    CL=interp1(AoA_list,CL_list,AoA,'pchip');
    CD=interp1(AoA_list,CD_list,AoA,'pchip');
    elseif ((radius_elem_midpoint(i)/R)>=0.30 &&
    (radius_elem_midpoint(i)/R)<0.35)
    load section2_AoA_Cl_Cd.dat;
    AoA_list=section2_AoA_Cl_Cd(:,1);
    CL_list=section2_AoA_Cl_Cd(:,2);
    CD_list=section2_AoA_Cl_Cd(:,3);
    CL=interp1(AoA_list,CL_list,AoA,'pchip');
    CD=interp1(AoA_list,CD_list,AoA,'pchip');
    elseif ((radius_elem_midpoint(i)/R)>=0.35 &&
    (radius_elem_midpoint(i)/R)<0.40)
    load section3_AoA_Cl_Cd.dat;
    AoA_list=section3_AoA_Cl_Cd(:,1);
    CL_list=section3_AoA_Cl_Cd(:,2);
    CD_list=section3_AoA_Cl_Cd(:,3);
    CL=interp1(AoA_list,CL_list,AoA,'pchip');
    CD=interp1(AoA_list,CD_list,AoA,'pchip');
    elseif ((radius_elem_midpoint(i)/R)>=0.40 &&
    (radius_elem_midpoint(i)/R)<0.45)
    load section4_AoA_Cl_Cd.dat;
    AoA_list=section4_AoA_Cl_Cd(:,1);
    CL_list=section4_AoA_Cl_Cd(:,2);
    CD_list=section4_AoA_Cl_Cd(:,3);
    CL=interp1(AoA_list,CL_list,AoA,'pchip');
    CD=interp1(AoA_list,CD_list,AoA,'pchip');
    elseif ((radius_elem_midpoint(i)/R)>=0.45 &&
    (radius_elem_midpoint(i)/R)<0.50)
    load section5_AoA_Cl_Cd.dat;
    AoA_list=section5_AoA_Cl_Cd(:,1);
    CL_list=section5_AoA_Cl_Cd(:,2);
    CD_list=section5_AoA_Cl_Cd(:,3);
    CL=interp1(AoA_list,CL_list,AoA,'pchip');
    CD=interp1(AoA_list,CD_list,AoA,'pchip');
    elseif ((radius_elem_midpoint(i)/R)>=0.50 &&
    (radius_elem_midpoint(i)/R)<0.55)
    load section6_AoA_Cl_Cd.dat;
    AoA_list=section6_AoA_Cl_Cd(:,1);
    CL_list=section6_AoA_Cl_Cd(:,2);
    CD_list=section6_AoA_Cl_Cd(:,3);
    CL=interp1(AoA_list,CL_list,AoA,'pchip');
    CD=interp1(AoA_list,CD_list,AoA,'pchip');
    elseif ((radius_elem_midpoint(i)/R)>=0.55 &&
    (radius_elem_midpoint(i)/R)<0.60)
    load section7_AoA_Cl_Cd.dat;
    AoA_list=section7_AoA_Cl_Cd(:,1);
    CL_list=section7_AoA_Cl_Cd(:,2);
    CD_list=section7_AoA_Cl_Cd(:,3);
    CL=interp1(AoA_list,CL_list,AoA,'pchip');
    CD=interp1(AoA_list,CD_list,AoA,'pchip');
    elseif ((radius_elem_midpoint(i)/R)>=0.60 &&
    (radius_elem_midpoint(i)/R)<0.65)

```



```

load section8_AoA_Cl_Cd.dat;
AoA_list=section8_AoA_Cl_Cd(:,1);
CL_list=section8_AoA_Cl_Cd(:,2);
CD_list=section8_AoA_Cl_Cd(:,3);
CL=interp1(AoA_list,CL_list,AoA,'pchip');
CD=interp1(AoA_list,CD_list,AoA,'pchip');
elseif ((radius_elem_midpoint(i)/R)>=0.65 &&
(radius_elem_midpoint(i)/R)<0.70)
load section9_AoA_Cl_Cd.dat;
AoA_list=section9_AoA_Cl_Cd(:,1);
CL_list=section9_AoA_Cl_Cd(:,2);
CD_list=section9_AoA_Cl_Cd(:,3);
CL=interp1(AoA_list,CL_list,AoA,'pchip');
CD=interp1(AoA_list,CD_list,AoA,'pchip');
    elseif ((radius_elem_midpoint(i)/R)>=0.70 &&
(radius_elem_midpoint(i)/R)<0.75)
load section10_AoA_Cl_Cd.dat;
AoA_list=section10_AoA_Cl_Cd(:,1);
CL_list=section10_AoA_Cl_Cd(:,2);
CD_list=section10_AoA_Cl_Cd(:,3);
CL=interp1(AoA_list,CL_list,AoA,'pchip');
CD=interp1(AoA_list,CD_list,AoA,'pchip');
elseif ((radius_elem_midpoint(i)/R)>=0.75 &&
(radius_elem_midpoint(i)/R)<0.80)
load section11_AoA_Cl_Cd.dat;
AoA_list=section11_AoA_Cl_Cd(:,1);
CL_list=section11_AoA_Cl_Cd(:,2);
CD_list=section11_AoA_Cl_Cd(:,3);
CL=interp1(AoA_list,CL_list,AoA,'pchip');
CD=interp1(AoA_list,CD_list,AoA,'pchip');
elseif ((radius_elem_midpoint(i)/R)>=0.80 &&
(radius_elem_midpoint(i)/R)<0.85)
load section12_AoA_Cl_Cd.dat;
AoA_list=section12_AoA_Cl_Cd(:,1);
CL_list=section12_AoA_Cl_Cd(:,2);
CD_list=section12_AoA_Cl_Cd(:,3);
CL=interp1(AoA_list,CL_list,AoA,'pchip');
CD=interp1(AoA_list,CD_list,AoA,'pchip');
elseif ((radius_elem_midpoint(i)/R)>=0.85 &&
(radius_elem_midpoint(i)/R)<0.90)
load section13_AoA_Cl_Cd.dat;
AoA_list=section13_AoA_Cl_Cd(:,1);
CL_list=section13_AoA_Cl_Cd(:,2);
CD_list=section13_AoA_Cl_Cd(:,3);
CL=interp1(AoA_list,CL_list,AoA,'pchip');
CD=interp1(AoA_list,CD_list,AoA,'pchip');
elseif ((radius_elem_midpoint(i)/R)>=0.90 &&
(radius_elem_midpoint(i)/R)<0.95)
load section14_AoA_Cl_Cd.dat;
AoA_list=section14_AoA_Cl_Cd(:,1);
CL_list=section14_AoA_Cl_Cd(:,2);
CD_list=section14_AoA_Cl_Cd(:,3);
CL=interp1(AoA_list,CL_list,AoA,'pchip');
CD=interp1(AoA_list,CD_list,AoA,'pchip');
elseif ((radius_elem_midpoint(i)/R)>=0.95 &&
(radius_elem_midpoint(i)/R)<1)
load section15_AoA_Cl_Cd.dat;
AoA_list=section15_AoA_Cl_Cd(:,1);
CL_list=section15_AoA_Cl_Cd(:,2);
CD_list=section15_AoA_Cl_Cd(:,3);
CL=interp1(AoA_list,CL_list,AoA,'pchip');

```

```

CD=interp1(AoA_list,CD_list,AoA,'pchip');
end
%Calculation of thrust force coefficient%
Cthrust=CL*cos(phi)+CD*sin(phi);
%Calculation of Torque force coefficient%
Ctorque=CL*sin(phi)-CD*cos(phi);
%Tip Loss Correction Factor
f_tip=0.5*Blades*((R-
radius_elem_midpoint(i))/(radius_elem_midpoint(i)*sin(phi)));
F_tip=(2/pi)*acos(exp(-1*f_tip));
%Hub Loss Correction Factor
f_hub=0.5*Blades*((radius_elem_midpoint(i)-
hubradius)/(radius_elem_midpoint(i)*sin(phi)));
F_hub=(2/pi)*acos(exp(-1*f_hub));
F=F_tip*F_hub;
if a<=0.4

a_new=1/(((8*pi*radius_elem_midpoint(i)*F*sin(phi)*sin(phi))/(chord_elem
_midpoint(i)*Blades*Cthrust))+1);
else
    %Buhl Model
    C_T=(8/9)+((4*F)-(40/9))*a+(((50/9)-(4*F))*(a^2));
    a_new=((18*F)-20-(3*((C_T*(50-(36*F)))+(12*F*((3*F)-
4))))^0.5)/((36*F)-50);
end
diff_btwn_a_and_a_new=abs(a-a_new);
a=0.5*(a+a_new);

a_dash=1/(((8*pi*radius_elem_midpoint(i)*F*sin(phi)*cos(phi))/(chord_ele
m_midpoint(i)*Blades*Ctorque))-1);
end
%Axial velocity just in front of turbine relative to windspeed
relative to wind speed
v=(1-a)*U_wind;
%circumferential velocity of the rotor relative to windspeed
u=(lamda_r*(1+a_dash))*U_wind;
%Relative Velocity%
U_rel=sqrt(v^2 + u^2);
%Pressure Drop%
Pressuredrop=density*(omega+(0.5*2*a_dash*omega))*(2*a_dash*omega)*(radi
us_elem_midpoint(i).^2);
%Elemental Thrust force%
Dthrust=0.5*density*Blades*U_rel*U_rel*chord_elem_midpoint(i)*Cthrust*dr
;
%Elemental Torque%
Dtorque=0.5*density*Blades*U_rel*U_rel*chord_elem_midpoint(i)*Ctorque*ra
dius_elem_midpoint(i)*dr;
TotalThrust=TotalThrust+Dthrust;
TotalTorque=TotalTorque+Dtorque;
fprintf(' %7.5f %7.5f %7.5f %7.3f %7.3f %7.4f
%7.4f %7.4f %7.4f %8.5f %8.6f %8.6f %8.5f
%8.5f %8.5f \n',
radius_elem_midpoint(i),chord_elem_midpoint(i),twist_elem_midpoint(i),ph
i,AoA,a,v,U_rel,CL,CD,Cthrust,Ctorque,Dthrust,Dtorque,Pressuredrop)
end
%Calculation of Power%
power = TotalTorque*omega;
fprintf(' Power = %5e Watts\n TotalTorque = %5e Nm \n TotalThrust = %5e
Newton \n',power,TotalTorque,TotalThrust)
% power coefficient
C_power = power/(0.5*density*(U_wind^3)*pi*R^2);

```

```
C_thrust= TotalThrust/(0.5*density*(U_wind^2)*pi*R^2);  
fprintf(' \n')  
fprintf(' C_power = %5f,    C_thrust = %5f \n' , C_power, C_thrust)
```

APPENDIX D: UDF for Neutral ABL at inlet of the domain

```
#include "udf.h"
#define znot 0.001
#define uref 10
#define zref 12
#define k 0.4
#define cmu 0.09
#define roughnessheight 0.05
DEFINE_PROFILE(normal_velocity, thread, position)
{
    #if !RP_HOST
    real x[ND_ND];
    real z;
    real zbyznot;
    real ufric;
    face_t f;
    ufric=(k*uref)/log(zref/znot);
    begin_f_loop(f, thread)
    {
        F_CENTROID(x,f,thread);
        z = x[2];
        zbyznot=z/znot;
        F_PROFILE(f,thread,position) = (ufric/0.4)*log(zbyznot);
    }
    end_f_loop(f,thread)
    #endif
}
DEFINE_PROFILE(k_value, thread, position)
{
    #if !RP_HOST
    real x[ND_ND];
    real z;
    real zbyznot;
    real logzbyznot;
    real ufric;
    face_t f;
    ufric=(k*uref)/log(zref/znot);
    begin_f_loop(f, thread)
    {
        F_CENTROID(x,f,thread);
        z = x[2];
        zbyznot=z/znot;
        logzbyznot=log(zbyznot);
        F_PROFILE(f,thread,position) = (pow(ufric,2))/(pow(cmu,0.5));
    }
    end_f_loop(f,thread)
    #endif
}
DEFINE_PROFILE(epsilon_value, thread, position)
{
    #if !RP_HOST
    real x[ND_ND];
    real z;
```

```

real zbyznot;
real logzbyznot;
real ufriC;
face_t f;
ufriC=(k*uref)/log(zref/znot);
begin_f_loop(f, thread)
{
F_CENTROID(x,f,thread);
z = x[2];
zbyznot=z/znot;
logzbyznot=log(zbyznot);
F_PROFILE(f,thread,position) =(pow(ufriC,3))/(k*(z));
}
end_f_loop(f,thread)
#endif
}
DEFINE_PROFILE(roughness_constant, thread, position)
{
#ifdef !RP_HOST
face_t f;
begin_f_loop(f, thread)
{
F_PROFILE(f,thread,position)=(9.793*znot/roughnessheight);
}
end_f_loop(f,thread)
#endif
}

```

APPENDIX E: NREL Phase VI blade geometry data

Radical Distance r (m)	Span Station ($r/5.029$)	Chord of the aerofoil at that particular section (m)	Twist (degrees)	Thickness (m)	Twist Axis (% chord)
0.0	0.0	Hub – center of rotation	Hub – center of rotation	Hub – center of rotation	Hub – center of rotation
0.508	0.101	0.218 (root hub adapter)	0.0(root hub adapter)	0.218	50 (root hub adapter)
0.660	0.131	0.218	0.0	0.218	50
0.883	0.176	0.183	0.0	0.183	50
1.008	0.200	0.349	6.7	0.163	35.9
1.067	0.212	0.441	9.9	0.154	33.5
1.133	0.225	0.544	13.4	0.154	31.9
1.257	0.250	0.737	20.040	0.154	30
1.343	0.267	0.728	18.074	20.95% chord	30
1.510	0.300	0.711	14.292	20.95% chord	30
1.648	0.328	0.697	11.909	20.95% chord	30
1.952	0.388	0.666	7.979	20.95% chord	30
2.257	0.449	0.636	5.308	20.95% chord	30
2.343	0.466	0.627	4.715	20.95% chord	30
2.562	0.509	0.605	3.425	20.95% chord	30
2.867	0.570	0.574	2.083	20.95% chord	30
3.172	0.631	0.543	1.150	20.95% chord	30
3.185	0.633	0.542	1.115	20.95% chord	30
3.476	0.691	0.512	0.494	20.95% chord	30
3.781	0.752	0.482	-0.015	20.95% chord	30
4.023	0.800	0.457	-0.381	20.95% chord	30
4.086	0.812	0.451	-0.475	20.95% chord	30
4.391	0.873	0.420	-0.920	20.95% chord	30
4.696	0.934	0.389	-1.352	20.95% chord	30
4.780	0.950	0.381	-1.469	20.95% chord	30
5.029	1	0.358	-1.775	20.95% chord	30

APPENDIX F: TURBINEinABL.jou (for $z_0=0.001\text{m}$)

```
/file/read-case"/gpfs/home/ijaz0001/opensea/innerdomain.msh"
/define/boundary-conditions/modify-zones/append-mesh"/gpfs/home/ijaz0001/opensea/outerdomain.msh"
/define/units/angular-velocity rpm
/define/models/viscous/ke-standard? yes
/define/models/viscous/curvature-correction? yes
/define/user-defined/interpreted-functions "/gpfs/home/ijaz0001/opensea/kepsilon.c" "cpp" 10000 no
/define/boundary-conditions/fluid new_innerdomain_blades_horizontal no no no yes -1 no 72 no 0 no 0
no 0 no 0 no 0 no 12 no 0 no -1 no 0 "none" no no no no no
/define/boundary-conditions/fluid created_material_12 no no no yes -1 no 0 no 0 no 0 no 0 no 0
no 12 no 0 no -1 no 0 "none" no no no no no
/define/boundary-conditions/velocity-inlet inlet no no yes yes yes yes "udf" "normal_velocity" no 0
yes yes yes "udf" "k_value" yes yes "udf" "epsilon_value"
/define/boundary-conditions/outflow outflow 1
/define/boundary-conditions/symmetry symmetry_farfield1
/define/boundary-conditions/velocity-inlet inlet_top yes yes no 11.519395 no 0 yes no 0 no 1 no 0
yes no 0.604534855 no 0.003861747778
/define/boundary-conditions/symmetry symmetry_farfield3
/define/boundary-conditions/wall nacalleandtower no no no 0 no 0.5
/define/boundary-conditions/wall ground no no no 0.05 yes yes "udf" "roughness_constant"
/define/boundary-conditions/wall bladesandhub yes motion-bc-moving no yes yes no no 0 no 0.5 0 0 0
12 0 -1 0
/define/mesh-interfaces/create inter1 interface1.1 () interface2.1 () no no no no
/define/mesh-interfaces/create inter2 interface1.2 () interface2.2 () no no no no
/define/mesh-interfaces/create inter3 interface1.3 () interface2.3 () no no no no
/solve/set/p-v-coupling 20
/solve/set/gradient-scheme no yes
/solve/set/discretization-scheme/pressure 12
/solve/set/discretization-scheme/mom 0
/solve/set/discretization-scheme/k 0
/solve/set/discretization-scheme/epsilon 0
/solve/set/limits 1 5e+10 1e-14 1e-20 1e+20
/solve/monitors/residual/check-convergence? yes yes yes yes yes yes
/solve/monitors/residual/convergence-criteria 1e-03 1e-06 1e-06 1e-05 1e-03 1e-03
/solve/monitors/residual/criterion-type 0
/define/models/unsteady-1st-order? yes
/define/boundary-conditions/fluid new_innerdomain_blades_horizontal no no no no 0 no 0 no 0 no 0
no 0 no 1 yes -1 no 72 no 0 no 0 no 0 no 0 no 12 no 0 no -1 no 0 "none" no no no no no
/define/boundary-conditions/fluid created_material_12 no no no no 0 no 0 no 0 no 0 no 0 no 1 yes
-1 no 0 no 0 no 0 no 0 no 0 no 12 no 0 no -1 no 0 "none" no no no no no
/solve/set/under-relaxation/pressure 0.3
/solve/set/under-relaxation/density 1
/solve/set/under-relaxation/body-force 1
/solve/set/under-relaxation/mom 0.4
/solve/set/under-relaxation/k 0.4
/solve/set/under-relaxation/epsilon 0.4
/solve/set/under-relaxation/turb-viscosity 0.4
/solve/initialize/compute-defaults velocity-inlet inlet
/solve/initialize/initialize-flow
/solve/set/time-step 0.83333333
/solve/set/number-of-time-steps 60
/solve/set/max-iterations-per-time-step 50000
/solve/dual-time-iterate 60 50000
/file/write-case"/gpfs/home/ijaz0001/opensea/completeslidingmesh_1storder_URF0.4"
/file/write-data"/gpfs/home/ijaz0001/opensea/completeslidingmesh_1storder_URF0.4"
/solve/set/time-step 0.00231481481481
/solve/set/number-of-time-steps 720
/solve/set/max-iterations-per-time-step 50000
/solve/dual-time-iterate 720 50000
/file/write-case"/gpfs/home/ijaz0001/opensea/completeslidingmesh_1STORDER720_URF0.4"
/file/write-data"/gpfs/home/ijaz0001/opensea/completeslidingmesh_1STORDER720_URF0.4"
/solve/set/under-relaxation/mom 0.7
/solve/set/under-relaxation/k 0.8
/solve/set/under-relaxation/epsilon 0.8
/solve/set/under-relaxation/turb-viscosity 1
/solve/set/time-step 0.00231481481481
/solve/set/number-of-time-steps 720
/solve/set/max-iterations-per-time-step 50000
/solve/dual-time-iterate 720 50000
/file/write-case"/gpfs/home/ijaz0001/opensea/completeslidingmesh_1STORDER720_DEFAULT"
/file/write-data"/gpfs/home/ijaz0001/opensea/completeslidingmesh_1STORDER720_DEFAULT"
/solve/set/discretization-scheme/mom 1
```

```

/solve/set/discretization-scheme/k 1
/solve/set/discretization-scheme/epsilon 0
/solve/monitors/residual/convergence-criteria 1e-03 1e-05 1e-05 1e-05 1e-03 1e-03
/solve/set/under-relaxation/pressure 0.3
/solve/set/under-relaxation/density 1
/solve/set/under-relaxation/body-force 1
/solve/set/under-relaxation/mom 0.4
/solve/set/under-relaxation/k 0.4
/solve/set/under-relaxation/epsilon 0.4
/solve/set/under-relaxation/turb-viscosity 0.4
/solve/set/time-step 0.00231481481481
/solve/set/number-of-time-steps 720
/solve/set/max-iterations-per-time-step 50000
/solve/dual-time-iterate 720 50000
/file/write-case-"/gpfs/home/ijaz0001/opensea/completeslidingmesh_2NDORDER720_URF0.4"
/file/write-data-"/gpfs/home/ijaz0001/opensea/completeslidingmesh_2NDORDER720_URF0.4"
/solve/set/under-relaxation/pressure 0.3
/solve/set/under-relaxation/density 1
/solve/set/under-relaxation/body-force 1
/solve/set/under-relaxation/mom 0.45
/solve/set/under-relaxation/k 0.45
/solve/set/under-relaxation/epsilon 0.45
/solve/set/under-relaxation/turb-viscosity 0.45
/solve/set/time-step 0.00231481481481
/solve/set/number-of-time-steps 720
/solve/set/max-iterations-per-time-step 50000
/solve/dual-time-iterate 720 50000
/file/write-case-"/gpfs/home/ijaz0001/opensea/completeslidingmesh_2NDORDER720_URF0.45"
/file/write-data-"/gpfs/home/ijaz0001/opensea/completeslidingmesh_2NDORDER720_URF0.45"
/solve/set/under-relaxation/pressure 0.3
/solve/set/under-relaxation/density 1
/solve/set/under-relaxation/body-force 1
/solve/set/under-relaxation/mom 0.7
/solve/set/under-relaxation/k 0.8
/solve/set/under-relaxation/epsilon 0.8
/solve/set/under-relaxation/turb-viscosity 1
/solve/set/time-step 0.00231481481481
/solve/set/number-of-time-steps 720
/solve/set/max-iterations-per-time-step 50000
/solve/dual-time-iterate 720 50000
/file/write-case-"/gpfs/home/ijaz0001/opensea/completeslidingmesh_2NDORDER720_DEFAULT"
/file/write-data-"/gpfs/home/ijaz0001/opensea/completeslidingmesh_2NDORDER720_DEFAULT"
/solve/monitors/residual/convergence-criteria 1e-03 1e-06 1e-06 1e-06 1e-03 1e-03
/solve/set/time-step 0.00231481481481
/solve/set/number-of-time-steps 720
/solve/set/max-iterations-per-time-step 50000
/solve/dual-time-iterate 720 50000
/file/write-
case-"/gpfs/home/ijaz0001/opensea/completeslidingmesh_2NDORDER720_DEFAULT_CONVER1EMINUS6"
/file/write-
data-"/gpfs/home/ijaz0001/opensea/completeslidingmesh_2NDORDER720_DEFAULT_CONVER1EMINUS6"
/solve/set/discretization-scheme/mom 1
/solve/set/discretization-scheme/k 1
/solve/set/discretization-scheme/epsilon 1
/solve/set/number-of-time-steps 720
/solve/set/max-iterations-per-time-step 50000
/solve/dual-time-iterate 720 50000
/file/write-
case-"/gpfs/home/ijaz0001/opensea/completeslidingmesh_720_2ndorder(epsilonalso)_URF_default"
/file/write-
data-"/gpfs/home/ijaz0001/opensea/completeslidingmesh_720_2ndorder(epsilonalso)_URF_default "

```