Simulations study of fluid-structure interaction of a wind turbine

Zheng, Shuai

2018

Zheng, S. (2018). Simulations study of fluid-structure interaction of a wind turbine. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/89427>

<https://doi.org/10.32657/10220/46249>

SIMULATIONS STUDY OF FLUID-STRUCTURE INTERACTION OF A WIND TURBINE

ZHENG SHUAI

SCHOOL OF MECHANICAL AND AEROSPACE ENGINEERING

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

2018

ACKNOWLEDGEMENTS

I would like to express my warmest and sincerest appreciation to my supervisor Associate Professor Chua Leok Poh and my co-supervisor Associate Professor Chai Gin Boay for their kind and patient guidance, keen insights, encouragements and friendship. Their knowledge and experience helped me focus on the right track and their encouragements give me a lot of motivation to overcome the difficulties on the way of research work. Their positive working attitude and modest personality set a very good example for me and will give a great influence on my academic work. It is a pleasure of me to pursue my Ph.D degree under their guidance.

I would like to thank Nanyang Technological University for offering me this valuable opportunity to pursue Ph.D. and granting me financial support.

I would like to thank my parents for a lifetime of unconditional support and endless love along the process of my academic journey. Lastly, I would especially like to thank my wife and daughter who have been supporting me unconditionally and continuously which makes it possible for me to fully concentrate on my academic work. Without their love and care, I could not finish this research work.

TABLE OF CONTENTS

TABLE OF CONTENTS.....	3
ABSTRACT	9
NOMENCLATURE	11
LIST OF FIGURES.....	15
LIST OF TABLES.....	22
CHAPTER 1 INTRODUCTION	23
1.1 Background and motivation	23
1.2 Research objective	26
1.3 Scope.....	28
1.4 Organization of the report.....	28
1.5 Chapter summary.....	30
CHAPTER 2 LITERATURE REVIEW	31
2.1 Introduction of wind turbine	31
2.1.1 Overview of wind energy	31
2.1.1.1 Global installed wind capacities and commercialization of the wind energy industry.....	32
2.1.1.2 Variety of wind energy application and examples	34
2.1.2 Wind turbine configuration	36
2.2 Wind turbine rotor design	42
2.2.1 Aerodynamics of airfoil.....	42
2.2.2 Blade design theory	52
2.2.2.1 Betz theory.....	52

2.2.2.2 Rotor Wake Theory	54
2.2.2.3 Blade Element Theory	56
2.2.2.4 Blade Momentum Theory	58
2.3 Fluid-structure interaction of wind turbine	61
2.3.1 Introduction	61
2.3.2 Basic theory of fluid-structure interaction	62
2.3.2.1 Theory description	62
2.3.2.2 Solving method	63
2.3.2.3 Interface data transmission	65
2.3.3 Governing Equations	66
2.3.4 Flow induced vibration on wind turbine blade	67
2.3.5 Blade natural frequency and vibration mode	69
2.3.6 Aeroelastic stability of wind turbine	71
2.3.7 Numerical simulation study of fluid-structure interaction of wind turbine	80
2.3.7.1 Introduction	80
2.3.7.2 Numerical method and uncertainties	81
2.3.8 Hybrid RANS/LES simulation method for wind turbine	83
2.3.8.1 Characteristic of turbulent flow	83
2.3.8.2 Comparison of RANS model and LES model	85
2.4 Chapter summary	89
CHAPTER 3 WIND TURBINE ROTOR DESIGN AND FSI NUMERICAL SIMULATION CONFIGURATION	90
3.1 Wind turbine rotor design	90
3.1.1 Wind turbine rotor parameters selection	90

3.1.2 Mathematical modeling of blade aerodynamic characteristic	93
3.2 Configurations of numerical simulation for two-way FSI on wind turbine	98
3.2.1 Simulation tool introduction.....	98
3.2.2 Flow domain configuration	99
3.2.2.1 Turbulence model selection.....	99
3.2.2.2 Turbulence model verification	101
3.2.2.3 Computational domain setup	105
3.2.2.4 Computational mesh setup	107
3.2.2.5 Mesh independence investigation	109
3.2.3 Mechanical structure configuration.....	110
3.2.4 Fluid-structure interaction configuration	112
3.3 Chapter summary.....	113
CHAPTER 4 WIND TURBINE ROTOR FSI NUMERICAL SIMULATION RESULTS ANALYSIS.....	114
4.1 Numerical simulation results for two-way FSI on wind turbine	114
4.2 Flow domain simulation results analysis	114
4.2.1 Velocity streamline	114
4.2.2 Velocity vector	119
4.2.3 Pressure contour	122
4.2.4 Pressure coefficient	134
4.2.5 Power coefficient.....	140
4.2.6 Torque coefficient	143

4.2.7 Thrust coefficient.....	144
4.3 Structural analysis results.....	146
4.3.1 Equivalent stress and total deformation.....	146
4.3.2 Natural frequency and vibration mode.....	148
4.3.3 Blade tip displacement	158
4.4 Chapter summary.....	160
CHAPTER 5 WIND TURBINE ROTOR AEROELASTIC STABILITY INVESTIGATION AND ANALYSIS.....	162
5.1 Aeroelasticity problem introduction	162
5.2 Studying approach and model configuration	162
5.3 Wind turbine blade airfoil section static aeroelastic problem	164
5.4 Wind turbine blade airfoil section aeroelastic stability analysis	166
5.4.1 Airfoil moving equations and natural frequency	166
5.4.2 Steady state aerodynamic model	167
5.4.3 Unsteady state aerodynamic model	173
5.4.4 Determination of airfoil section parameters	174
5.4.4.1 Area of airfoil section	175
5.4.4.2 Mass center of airfoil section	175
5.4.4.3 Polar inertia moment to mass center.....	175
5.4.4.4 Moment inertia of airfoil section	176
5.4.4.5 Stiffness of vertical and twist spring	176
5.4.4.6 δ and e	176
5.4.5 Aerodynamic characteristics of airfoil section	176
5.5 Aeroelastic stability assessment and parameter analysis.....	180

5.5.1 Introduction of aeroelastic calculation	180
5.5.2 Mathematical approach.....	181
5.5.3 Cases study and results analysis	182
5.5.3.1 Blade aeroelastic stability when $\lambda=3$	183
5.5.3.2 Blade aeroelastic stability when $\lambda=2.4$	184
5.5.3.3 Blade aeroelastic stability when $\lambda=1.7$	186
5.5.3.4 Blade aeroelastic stability when $\lambda=1.5$	188
5.5.3.5 Phenomenon of frequency overlap.....	190
5.5.3.6 Influence of mass center location on aeroelastic stability	191
5.5.3.7 Impact of different wind turbine materials on aeroelastic stability	193
5.6 Chapter summary.....	194
CHAPTER 6 MULTIPLE OBJECTIVES OPTIMIZATION OF WIND TURBINE PERFORMANCE	196
6.1 Purpose of optimization	196
6.2 Multiple objectives optimization.....	196
6.3 Genetic algorithm.....	198
6.4 Wind turbine blade optimization	199
6.4.1 Define target function	199
6.4.1.1 Aerodynamic performance function	199
6.4.1.2 Blade loading function	200
6.4.1.3 Optimization model function	200
6.4.1.4 Constrains	201
6.4.1.5 Optimization flow chart	201

6.4.2 Optimization results and discussion	202
6.5 Chapter summary	211
CHAPTER 7 CONCLUSIONS AND FUTURE WORKS	212
7.1 Conclusions	212
7.2 Future works	214
REFERENCES.....	216
APPENDIXES.....	226
Appendix A	226
Appendix B	228
Appendix C	229
PUBLICATIONS	230

ABSTRACT

This report presents the study of fluid-structure interaction (FSI) of a 5kW horizontal axis wind turbines (HAWT). The motivation of the current investigation is to address the FSI problem of HAWT in the whole flow field analysis with hybrid turbulence model method which is used to find out the flow and structure characteristics of wind turbine in operation under different tip speed ratio (TSR). The target of current work is to obtain detailed information of flow field and structure response simultaneously using the finite element analysis (FEA) method as well as providing new reliable and accurate approach which could be quickly and economically employed in wind turbine FSI problem analysis.

In this report, a detailed and comprehensive literature review related to FSI problem of wind turbine has been presented and includes aerodynamic, wind turbine theory, computational fluid dynamic, fluid structure interaction and aeroelastic stability. A 5kW wind turbine is designed according to blade element momentum (BEM) method and a 3D wind turbine modeling is generated in SolidWorks. The 3D wind turbine modeling is imported into FEA software by coupling with the structure domain and the fluid domain to achieve FSI analysis and different TSR are applied to the computational fluid dynamics (CFD) analysis to gain the wind turbine operation information under different scenarios. The simulation results are investigated to reveal the influence of the FSI phenomenon on wind turbine performance. Analysis of wind turbine aeroelastic stability is conducted to prevent blade from experiencing flutter problem. A multiple objective optimization method for wind turbine design including aerodynamic and structure has been developed to improve wind turbine performance.

The flow field analysis results showed that the strength of vortex at the tip and root of the blade increases as wind speed increases when wind turbine is under operating conditions. With wind speed of 10m/s, most of the flow over the lower surface remains attached and the

wind turbine blade was running under design condition. As the wind speed increases, flow separation emerges. When the wind speed increasing to 16m/s, the wind turbine blade works under deep stall condition. The wind turbine power coefficient drops continuously from 0.46 to 0.22, as wind speed increased from 10m/s to 20m/s. The comparison of wind turbine power and torque coefficient under different CFD configuration shows the advantage of whole flow field analysis with hybrid turbulence model.

The structural analysis results demonstrated that most of the pressure force acts at a region around the 1/4 chord from the leading edge of the blade under operating condition and the displacement of blade tip in flap direction is bigger than those in the other directions. Modal analysis results deduced that flap is the main type of vibration in first and second modes. The difference of natural frequency between single blade and blade rotor becomes larger as vibration order increases. The dynamic performances of wind turbine blade and rotor are totally different and show the whole flow field method is better than the one third domain method. The wind turbine aeroelastic analysis results illustrate that the bending frequency and torsion frequency are around 23.94Hz and 38.46Hz respectively under TSR=4 and both move towards 30Hz under TSR=1.7. When TSR is smaller than 1.7 the wind turbine will experience flutter due to the overlapping of bending and torsion frequencies. The multiple objectives optimization method takes the aerodynamic performance and blade loading into account to gain better design parameter. The comparison between initial and optimal design shows that optimized method could provide higher electrical energy around 1073kW•h per annum under rated wind speed. The optimized wind turbine critical flutter wind speed will increase from initial design speed of 35m/s to 38m/s under optimization design.

Keywords: Wind Turbine; Computational Fluid Dynamics; Fluid-structure Interaction; Aeroelastic Stability, Multiple-objectives Optimization

NOMENCLATURE

Alphabet

a	axial induction factor
b	rotational induction factor
c	chord (m)
C_l	lift coefficient
C_d	drag coefficient
C_p	power coefficient
D	drag force (N)
e	distance of mass and torsion center (m)
E	Young's modulus (Pa)
f_B	body force vector (N)
f_{st}	velocity strain vector
F	force on blade(N)
F_{tip}	tip loss factor
F_T	total loss factor
F_{hub}	hub loss factor
F_x	force in x direction (N)
F_y	force in y direction (N)
I	moment of inertia (kg·m ²)
k	torsion stiffness (N·m/rad)
L	lift force (N)
m	mass (kg)
M	moment (N·m)

P	power output of wind turbine (W)
P_{ele}	power of the blade element (W)
P_{max}	maximum power for an ideal wind turbine (W)
P'	pressure drop (Pa)
q	heat flux tensor (W/m ²)
q^B	heat per unit volume (W/m ³)
$q(t)$	pattern of vibration
r_{hub}	radius of hub (m)
R	radius of rotor (m)
S	area of rotor (m ²)
s	bending stiffness (N/m)
S_1	area of actuator disk before rotor (m ²)
S_2	area of actuator disk after rotor (m ²)
T	thrust on wind turbine (N)
U_i	tensor
U_T	tangential velocity (m/s)
v	velocity vector (m/s)
$\hat{v}$	fluid velocity in ALE frame (m/s)
V	wind speed after rotor (m/s)
V_1	incoming wind speed (m/s)
V_2	far downstream wind speed (m/s)
V_t	induced tangential velocity (m/s)
x	displacement (m)

Greek Symbols

α	angle of attack (rad)
$\gamma(x)$	vibration mode
δ	distance of mass and aerodynamic center (m)
δ_{ij}	Kronecker delta
θ	twist angle (rad)
k	turbulent kinetic energy (J)
λ	tip speed ratio
λ_r	local tip speed ratio
μ	dynamic eddy viscosity (kg/m · s)
τ	fluid stress tensor (Pa)
$\emptyset$	wind direction angle (rad)
ω	rotor angular velocity (rad/s)
ω_n	natural frequency (Hz)
ω_T	turbulent frequency (Hz)
Ω	induced angular velocity (rad/s)

Others

D/Dt	time derivative in ALE frame
$(\nabla \cdot)$	divergence operator
∇	gradient operator
∇^2	Laplace operator
$(\cdot)$	dot product

List of Abbreviations

2D	two-dimensional
3D	three-dimensional
ALE	Arbitrary Lagrangian-Eulerian
BEM	blade element momentum
CFD	computational fluid dynamics
FEA	finite element analysis
FFT	fast fourier transform
FSI	fluid-structure interaction
HAWT	horizontal axis wind turbines
LES	large eddy simulation
NASA	National Aeronautics and Space Administration
NREL	National Renewable Energy Laboratory
PIV	particle image velocimetry
PSD	power spectrum density
RANS	Reynolds-averaged Navier-Stokes
TSR	tip speed ratio
VAWT	vertical axis wind turbines

LIST OF FIGURES

Figure 2.1 (a) global annual and (b) cumulative installed wind capacities from 1997 to 2016 (GWEC 2016).....	33
Figure 2.2 Percentage of new installed wind capacity in different countries in 2016 (GWEC 2016).....	34
Figure 2.3 (a) single-mast and (b) multiple-mast sailing ship (c) Holland wind mill and (d) American wind mill	36
Figure 2.4 Different types of Darrieus vertical axis wind turbines	37
Figure 2.5 (a) vertical axis wind turbine and (b) horizontal axis wind turbine (Manwell et al. 2002)	38
Figure 2.6 Two kinds of rotor configurations: (a) upwind and (b) downwind ...	39
Figure 2.7 Power coefficient versus tip speed ratio curve (Burton et al. 2011)	41
Figure 2.8 Maximum achievable power coefficients as a function of number of blades (Burton et al. 2011).....	41
Figure 2.9 Definition of lift, drag, angle of attack, chord and aerodynamic moment	43
Figure 2.10 C_l vs C_d curve of airfoil (a) S822 (Selig and Granahan 2004) and (b) S823 (Selig et al. 1995) under different Reynolds number	45
Figure 2.11 Variation of maximum power coefficient for different lift to drag ratios.....	46
Figure 2.12 Transition of flow from laminar to turbulent along upper surface of airfoil	47
Figure 2.13 Profile of airfoil boundary layer (a) favourable pressure gradient and (b) adverse pressure gradient	48
Figure 2.14 Rotor of wind turbine	51
Figure 2.15 Velocity analysis of section dr	51
Figure 2.16 Energy extracting actuator and annular stream tube	52

Figure 2.17 Different kind of vortex behind wind turbine	55
Figure 2.18 Local force and velocity on a wind turbine blade element at radius r	57
Figure 2.19 Flow chart of wind turbine blade design process	61
Figure 2.20 FSI calculation process chart	63
Figure 2.21 Comparison of two kinds FSI interface dealing techniques (a) interface tracking technique and (b) interface capturing technique	65
Figure 2.22 Three main kinds of blade deformation: (a) flap, (b) swing and (c) twist.....	68
Figure 2.23 Cantilever beam natural frequency and vibration mode (a) cantilever beam configuration, (b) first order vibration mode and frequency, (c) second order vibration mode and frequency and (d) third order vibration mode and frequency.....	71
Figure 2.24 Triangle of aeroelastic force	72
Figure 2.25 Energy obtained in airfoil section within one vibration period (a) zero degree phase difference and (b) 90 degree phase difference	75
Figure 2.26 Aeroelastic system flow chart (a) structure dynamic subsystem, (b) aerodynamic subsystem and (c) feedback coupling of two subsystems.....	76
Figure 2.27 Resonance diagram of rotor blade under flow induced deformation (a) 1st flapwise vibration and (b) 1st edgewise vibration	79
Figure 2.28 Energy cascade of flow field	85
Figure 2.29 Classification of computational fluid dynamic turbulence model ..	87
Figure 2.30 Vortex shedding from cylinder using different turbulence method (a) URANS (b) SAS (Ansys 2015)	88
Figure 3.1 Twist angle distributions	96
Figure 3.2 Chord length distributions.....	96

Figure 3.3 Blade element twist angle variation (from root to tip)	97
Figure 3.4 Isometric view for wind turbine blade.....	97
Figure 3.5 Isometric view for wind turbine rotor with hub	98
Figure 3.6 (a) Wind turbine installed in wind tunnel (b) photo of wake flow visualization (Hand et al. 2001)	102
Figure 3.7 Comparison of the present simulation results with experimental data of C_p (a) 7m/s and (b) 25m/s at location of $r/R=0.8$, (c) 7m/s and (d) 25m/s at location of $r/R=0.46$	104
Figure 3.8 Flow field domain and boundary conditions	106
Figure 3.9 Mesh growth rate around the rotor surface is 1.1 in flow domain	107
Figure 3.10 Inflation layers around the wind turbine blade in flow domain...	108
Figure 3.11 Blade mesh around tip area in structure domain	111
Figure 3.12 Rotor mesh around root area in structure domain.....	111
Figure 3.13 Process flow chart of two-way fluid-structure interaction.....	112
 Figure 4.1 Velocity streamline for wind speed 13m/s	115
Figure 4.2 Streamline distributions on lower surface at wind speeds of (a) 10m/s, (b) 13m/s, (c) 16m/s and (d) 20m/s. Note that wind velocity V , flows out from the paper.....	117
Figure 4.3 Streamline distributions on upper surface of wind turbine blade: (a) 10m/s, (b) 13m/s, (c) 16m/s, (d) 20m/s, flows out from the paper.....	118
Figure 4.4 Enlarge view of stagnation point of streamline under wind speed of 10m/s.....	119
Figure 4.5 Velocity vectors on lower surface with wind speeds of (a) 10m/s, (b) 13m/s, (c) 16m/s and (d) 20m/s. Note that the turbine blade is rotating in clockwise direction.	122

Figure 4.6 Pressure contour at different locations (a) $r/R=0.25$, (b) $r/R=0.50$, (c) $r/R=0.75$ and (d) $r/R=0.95$ of a wind turbine blade under wind speed of 10m/s	125
Figure 4.7 Pressure contour at different locations (a) $r/R=0.25$, (b) $r/R=0.50$, (c) $r/R=0.75$ and (d) $r/R=0.95$ of a wind turbine blade under wind speed of 16m/s	127
Figure 4.8 Pressure contour at different locations (a) $r/R=0.25$, (b) $r/R=0.50$, (c) $r/R=0.75$ and (d) $r/R=0.95$ of a wind turbine blade under wind speed of 20m/s	130
Figure 4.9 Pressure contour at locations $r/R=0.95$ of wind turbine blade under wind speed of 20m/s at time (a) 0.02s, (b) 0.05s, (c) 0.08s and (d) 0.1s	133
Figure 4.10 Pressure coefficients at six locations (a) $r/R=0.40$, (b) $r/R=0.50$, (c) $r/R=0.60$, (d) $r/R=0.70$ (e) $r/R=0.80$ (f) $r/R=0.90$ of wind turbine blade under wind speed 10m/s.....	135
Figure 4.11 Pressure coefficients at six locations (a) $r/R=0.40$, (b) $r/R=0.50$, (c) $r/R=0.60$, (d) $r/R=0.70$ (e) $r/R=0.80$ (f) $r/R=0.90$ of wind turbine blade under wind speed 16m/s.....	137
Figure 4.12 Pressure coefficients at location $r/R=0.80$ of wind turbine blade under wind speed 16m/s at different time (a) 0.02s, (b) 0.04s, (c) 0.05s, (d) 0.07s, (e) 0.09s and (f) 0.10s.....	139
Figure 4.13 Wind turbine power coefficient versus different wind speeds	141
Figure 4.14 Wind turbine rotor power coefficient versus tip speed ratio under different flow domain configuration and turbulence model	143
Figure 4.15 Wind turbine torque coefficient versus tip speed ratio under different flow domain configuration and turbulence model	144
Figure 4.16 Thrust coefficient distribution of wind turbine rotor at different tip speed ratios	145

Figure 4.17 Equivalent Stress of wind turbine rotor. Note that the rotor is rotating in anticlockwise direction and the stress unit is Pa.....	147
Figure 4.18 Total deformation of wind turbine rotor. Note that the rotor is rotating in anticlockwise direction and the deformation unit is m.....	147
Figure 4.19 Vibration modes of turbine blade at wind velocity of 10m/s (a) First order, (b) Second order, (c) Third order, (d) Forth order, (e) Fifth order and (f) Sixth order	149
Figure 4.20 Vibration modes of turbine rotor at wind velocity of 10m/s (a) First order, (b) Second order, (c) Third order, (d) Forth order, (e) Fifth order, (f) Sixth order, (g) Seventh order, (h)Eighty order and (i) Ninth order.....	155
Figure 4.21 Resonance diagram of a single blade and the whole wind turbine rotor with first three vibration orders.....	158
Figure 4.22 Displacement of tip part of blade during wind turbine operation in (a) X direction, (b) Y direction and (c) Z direction (the direction of X, Y and Z refer to coordinate system shown in Figure 4.21)	160
Figure 5.1 (a) A typical airfoil section model and (b) displacement coordinates system (Dowell et al. 2005).....	163
Figure 5.2 Lift force and drag force on airfoil and projection direction	168
Figure 5.3 Airfoil section velocity triangle (a) no vibration and (b) with vibration	169
Figure 5.4 Airfoil coordinate systems at leading edge and mass center	175
Figure 5.5 The real part of the eigen value of the airfoil in vertical direction versus tip speed ratio at $r/R=0.5$	177
Figure 5.6 The real part of the eigen value of the airfoil in twist direction versus tip speed ratio at $r/R=0.5$	178

Figure 5.7 Lift and drag coefficient comparison between experimental data and CFD data of airfoil S822 (a) lift and (b) drag	179
Figure 5.8 Angle of attack range extension of lift and drag coefficient	180
Figure 5.9 Airfoil section aeroelastic stability analysis when tip speed ratio, $\lambda=3$ (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane	184
Figure 5.10 Airfoil section aeroelastic stability analysis when tip speed ratio, $\lambda=2.4$ (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane.....	186
Figure 5.11 Airfoil section aeroelastic stability analysis when tip speed ratio, $\lambda=1.7$ (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane.....	188
Figure 5.12 Airfoil section aeroelastic stability analysis when tip speed ratio, $\lambda=1.5$ (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane.....	190
Figure 5.13 Bending and torsion vibration frequency move towards overlap as the increasement of wind speed.....	191
Figure 5.14 Impact of mass center location on the flutter wind speed	192
Figure 5.15 Real part of eigen values in bending direction of three differential wind turbine blade materials (a) fiber glass, (b) aluminum alloy and (c) steel under differential tip speed ratios	194
Figure 6.1 Flow chart of optimization program with genetic algorithm	202

Figure 6.2 Comparison of chord and twist angle between initial design and optimized design (a) chord and (b) twist angle	204
Figure 6.3 Comparison of wind turbine performance between initial design and optimized design (a) power coefficient, (b) torque coefficient and (c) thrust coefficient.....	207
Figure 6.4 Airfoil section aeroelastic stability analysis when $\lambda=1.7$ with optimized blade design (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane	209
Figure 6.5 Airfoil section aeroelastic stability analysis when $\lambda=1.6$ with optimized blade design (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane	210

LIST OF TABLES

Table 3.1 NREL airfoil family (Tangler and Somers 1996)	93
Table 3.2 Part of cross section data from NREL Phase VI experiment	101
Table 3.3 Verification of mesh independence	110
Table 4.1 Vibration frequency comparison of single blade and whole rotor ..	156

CHAPTER 1 INTRODUCTION

1.1 Background and motivation

A wind turbine is a rotating machine which converts the kinetic energy absorbed from wind to mechanical energy. Based on the orientation of axis, the wind turbines can be classified into two categories: horizontal axis wind turbines (HAWT) and vertical axis wind turbines (VAWT). Nowadays, the majority of commercial wind turbines in operation are HAWT because of their high efficiency, stable performance and relatively low cost (Jamieson 2011). The present work will thus focus on HAWT. The main motivation for using wind turbine to generate electricity is low CO₂ emission over its entire life span which can help in slowing down the global warming from bad to worse. Wind energy also provides an alternative option to the fossil fuel, which can dramatically reduce the dependence of its import of a nation. These merits make wind energy a suitable renewable energy to support the sustainable development of the country especially those lack of natural fuel energy resource (GWEC 2015). The advantages of wind turbine are very important to an island country like Singapore which wants to move towards a self-sustainable development society.

Wind turbine blade design and operation should aim for high power efficiency and long life span. Currently the most widely employed wind turbine blade design method is the improved blade element momentum method (BEM) which includes the tip and hub loss coefficients. The BEM method is derived based on the mass conservation law, momentum conservation law and energy conservation law (Hansen 2008). This method assumes that the wind turbine blade is operating under steady state and the flow field on each blade element does not interact with the other element. There are three main kinds of loads applied on wind turbine blade: the gravitational load, inertial load and aerodynamic load. These loads change with time and interact with wind turbine blade which is known as fluid structure interaction (FSI)

and has great influence on the wind turbine aerodynamic and structure performance. Therefore, it is very important to investigate and analyze the behavior of FSI of wind turbine (Hau 2013).

The FSI effect is very crucial to wind turbine design and operation but it is very difficult to carry out the analysis. Wind turbine rotors often operate at large angles of attack in the deep stall regime, which results in strong and large-scale flow separation and viscous effects. Under such conditions, leading-edge vortices grow and subsequently shed into the near wake region. The flow field simultaneously contains laminar, transition and multiple turbulent flow regimes, which make dynamic load and FSI difficult to determine, especially when using three-dimensional (3D) calculation method (Takizawa and Tezduyar 2011). Therefore, flow around wind turbine blades as well as in the wake region are multi-scale and multi-physics in nature. Scale variations, from the length of the blades to the smallest turbulent eddies, are enormous. The blades interact nonlinearly with the complex air flow, exhibiting flutter and edge-wise vibration and nonlinear aero-elastic bifurcation, adding complexity to the overall problem (Chaviaropoulos et al. 2003). The aerodynamic and structural dynamic optimization of turbine blade is a highly complicated and time-consuming process, and can only be performed through trial and error by experienced designers, therefore, new generations of turbines take a very long time to reach market. To minimise the computational difficulties, a four-step approach to solution is proposed.

Firstly, in routine engineering analysis and design, engineers can only study a 120 degree segment of the flow field around the rotor with symmetrical condition is being assumed, which means that the flow field has to be assumed in steady state or at least periodic boundary conditions have to be used (Chaviaropoulos et al. 2003). Furthermore, the flow around the blade and the wake regions are separately simulated, resulting inaccuracy in the prediction of wind turbine power performance. Therefore, the whole flow field research

method which could provide more reliable and accurate simulation results are being developed in the present work.

Secondly, the flow around the rotor and in the wake is fully three-dimensional and unsteady in nature which cannot be artificially separated (Rasmussen et al. 2003). Therefore, the large eddy simulation (LES) is required to capture all the details of flow field. The problem is that even for just a single blade simulation and does not mention the whole flow field investigation, the LES simulation is still required excessive resources and computing time which make this turbulence model could not be easily and conveniently applied in general design and analysis work. On the other hand, due to the multi-scale of the flow around wind turbine the commonly used Reynolds Average Navier-Stokes (RANS) turbulence model such as $k - \varepsilon$ or $k - \omega$ which could be easily deployed is not capable to deal with these circumstances as well as delivering a trustable and precise result (Sagaut et al. 2013). In the current work, a new approach of aerodynamic analysis: the whole flow field analysis with the FSI effect is taking into account and the using of the hybrid RANS/LES turbulence model is proposed.

Thirdly, the aeroelastic stability is another aspect of common FSI problem for modern high speed wind turbine due to the long blade and relatively low stiffness of material as well as the high blade rotating speed (Zhang and Huang 2011). To precisely predict the aeroelastic performance of wind turbine blade under several working conditions with different aerodynamic model, the accurate data of airfoil lift and drag coefficients is a key factor. The hybrid RANS/LES turbulence model could deliver reliable and accurate airfoil data especially when the wind turbine blade is under high angle of attack which is the situation when blade encounters aeroelastic problem. This method provides wind turbine engineer a new approach to quickly and precisely verify the aeroelastic stability of wind turbine blade with an economical price.

Fourthly, current design optimization methods are based on simplified theory such as Blade Element Momentum (BEM) theory which is not optimal for the wind turbine aerodynamic performance and does not include the blade loads analysis (Brondsted and Nijssen 2013). The BEM method could not provide a very accurate design result. It is necessary to develop an improved method by considering all factors mentioned above to support the engineer to design a wind turbine with high efficiency, stable performance and long life span.

Using the new whole flow field and hybrid turbulence model analysis approach, it is able to overcome all the weaknesses of the current engineering analysis of wind turbines and have a better understanding of the true flow physics. The computational results of FSI on wind turbine blade as well as the whole rotor will then be able fully and deeply analyzed to help researchers understand the physics of FSI phenomenon. The new calculation and optimization method proposed in this work will also provide the wind energy industry an economical and fast approach to design new wind turbine and verify the wind turbine performance. The novelty of this research work is to establish a new whole flow field with hybrid RANS/LES method to investigate the FSI of a wind turbine as well as develop a multiple objectives optimization method to improve the wind turbine power efficiency and life span.

1.2 Research objective

The objectives of the current work are therefore, as following,

(1) The 5kW wind turbine will be designed based on the BEM method and the 3D model of the wind turbine will be generated in software. The applicability of the hybrid turbulence model which is going to employ for the wind turbine numerical simulation will be verified according to the experimental data. The configuration of the whole flow field two-way FSI simulation will be conducted in the present study.

(2) The simulation results of the whole flow field two-way FSI of wind turbine will be analyzed. The flow domain analysis results including the flow separation, pressure contours and pressure coefficients to investigate the flow field around wind turbine blade under different wind speeds. The different simulation results of wind turbine power and torque coefficients with different CFD methods are used to verify that the whole flow field method with hybrid turbulence model and show that they could deliver accurate results. The Structure domain analysis results including vibration frequency and mode will be used to validate that using the whole flow field method will provide more realistic results than using one third domain method. The displacement of the blade and the aerodynamic force on the blade will further be obtained to investigate the behavior of the wind turbine blade under two-way FSI simulation.

(3) The physical model and equations of aeroelasticity of 5kW wind turbine blade will be developed to investigate the blade aeroelastic stability under different wind speeds. The lift and drag coefficients of airfoil S822 under a wide range of angles of attack will be obtained with hybrid turbulence model. The flutter wind speed of the blade will be obtained by analyzing the displacement, velocity and acceleration of blade section under different wind speeds. The physical explanation of the flutter phenomenon will be illustrated and the influencing factors including mass center and different materials on the incensement of the critical flutter wind speed will be analyzed too.

(4) A physics-based multi-disciplinary optimization method which could improve the aerodynamic shape and power efficiency of the wind turbine blade will be developed. This optimization method will include the parameters such as the chord, twist angle and aerodynamic force. The simulation results such as power coefficients, torque coefficients and critical flutter wind speed under initial design and optimized design will be compared to illustrate the improvement of performance of the wind turbine under optimization method.

1.3 Scope

The main scope of the present study is the FSI of the whole wind turbine blades and rotor. To achieve this target, the followings have been done. Firstly, detailed literatures review of aerodynamic, wind turbine, computational fluid dynamic, fluid structure interaction and aeroelastic stability. Secondly, a 5kW wind turbine is designed based on BEM method and a 3D modeling of wind turbine rotor is generated in SolidWorks. The configuration of flow and structure domain as well as the fluid structure interaction have been done in Ansys software. Thirdly, the simulation results are analyzed to illustrate the influence of the fluid structure interaction on wind turbine performance. Fourthly, investigation process of wind turbine aeroelastic stability is shown to keep the blade from flutter problem. Fifthly, a multiple objectives optimization method including the aerodynamic and structure performance has been developed to improve the wind turbine power efficiency and life span.

1.4 Organization of the report

In Chapter One, the background and motivation, the objectives, the scope and the organization of the report are presented. The four detailed objectives on the FSI of wind turbine rotor are outlined.

Literature review in the area of wind turbine FSI and the related basic concept and fundamentals are introduced in Chapter Two. The literature review includes five parts: introduction of wind turbine, aerodynamics, fluid-structure interaction, computational fluid dynamics and aeroelastic stability. These five parts are presented in sequences so as to illustrate the current status in FSI of wind turbine, the challenge of wind turbine design engineer encountered and the accessible innovation approach.

The parameters for the present 5kW wind turbine will be calculated based on the BEM method and a computer program will be employed to conduct the calculation in Chapter

Three. Configurations of numerical simulation for two-way FSI on wind turbine will be implemented in flow domain and structure domain separately. The computational results of the present hybrid turbulent design model will be verified according to the experimental data. The computational mesh will be generated and the mesh independency verification will be conducted. The parameters of fluid structure interaction such as time interval and converge criteria will be selected in the chapter.

The numerical simulation results will be presented in Chapter Four. The FSI numerical simulation is conducted under several different wind speed conditions. The flow field of numerical simulation is analyzed and the detailed information of flow such as velocity vectors, velocity streamlines and pressure contours will be illustrated and analyzed. The comparison of wind turbine powers, torques and thrust coefficients under different CFD configurations have been conducted to show the advantage of whole flow field analysis with hybrid method. The blade structural parameters (e.g. deformation and stress induced) determined from numerical simulation are used to provide accurate solution. The important parameters such as resonance frequency and the frequency differences between single blade and whole rotor under pressure load will be demonstrated and analyzed.

The aeroelasticity problem, research approach and model configuration are introduced in Chapter Five. The definitions of aerodynamic damping and stiffness as well as different aerodynamic models are illustrated. The aerodynamic characteristics of airfoil section is presented and the lift and drag coefficient curves of airfoil are extended to full wind range using the hybrid turbulence method. The aeroelastic case and results under different tip speed ratio are analyzed and the phenomenon of frequency overlap are being observed. The critical flutter wind speed has been detected in order to prevent the wind turbine blade experience flutter issue.

The multiple objectives optimization method and the genetic algorithm of the wind turbine optimization process will be presented in Chapter Six. In the designing stage of the optimization method the aerodynamic performance and blade loading are taken into account to gain the optimal wind turbine design. The comparison of power performance and aeroelastic stability with initial design and optimal design are conducted. The comparison of results show that the optimal design based on the multiple objectives optimization could improve wind turbine power performance up to 4%, torque performance up to 4% and increase critical flutter wind speed from 35m/s to 38m/s.

The study will be concluded and the future works will be proposed in the last chapter.

1.5 Chapter summary

In this Chapter, the background and motivation of present research are introduced. The research objectives of the current work are illustrated in details. The organization of the report includes the following sections: the literature review of wind turbine, 5kW wind turbine design, wind turbine FSI numerical simulation, aeroelasticity problem analysis, multiple objectives optimization and future works.

CHAPTER 2 LITERATURE REVIEW

2.1 Introduction of wind turbine

2.1.1 Overview of wind energy

A modern wind turbine is a machine which converts the wind power into electricity and acts as the electricity generator in power network. The modern wind turbine has started to appear when Betz applied the actuator disc theory to the wind turbine and discovered that a maximum of 59.3% of the kinetic wind energy can be converted into mechanical energy by a free-stream wind turbine in 1920 (Gasch and Tewele 2012). After the Betz limit theory has established, the development of wind turbine has been accelerated and many promising approaches for modern wind turbine design have emerged. A wind turbine which converts the kinetic energy absorbed from wind to mechanical energy can be classified into two categories: horizontal axis wind turbine (HAWT) and vertical axis wind turbine (VAWT). Currently, the majority of commercial wind turbines in operation are HAWT because of their high efficiency, stable performance and relatively low cost (Jamieson 2011). The present chapter will focus on HAWT. Furthermore, the main motivation for using wind turbines to generate electricity is low CO₂ emission which can help in slowing down the rate of global warming. These merits make wind energy a suitable renewable energy to support the sustainable development of all countries, especially those lack of natural fuel energy resources such as island country like Singapore (GWEC 2016).

The wind turbine blade is the most important component which is used to extract energy from wind. The blade design and operation should thus be targeted for high power efficiency and long life span. Currently the wind turbine blade is designed according to the improved blade element momentum method (BEM) after taking the tip and hub loss coefficients into consideration. The BEM method is derived from the mass conservation law, momentum

conservation law and energy conservation law (Hansen 2008). This method assumes that the wind turbine blade is operating under steady state and the flow field on each blade element does not interact with the other element. In the real situation, there are three main loads applied on a wind turbine blade: the gravitational load, inertial load and aerodynamic load. All the three loads applied on each wind turbine blade vary with time. The change in load with time will affect the deformation of each wind turbine blade, which, in turn, will affect the flow field and the wake around the blade, thus resulting in a continuous variation of the load. This kind of two-way coupling between the blade and flow is known as fluid-structure interaction (FSI) (Wang 2008). The FSI of wind turbine blades will affect the power generating efficiency, stability and life span of a wind turbine. The current chapter will focus on a detailed analysis of the behavior of FSI of wind turbines.

2.1.1.1 Global installed wind capacities and commercialization of the wind energy industry

According to the Global Wind Report 2016 which was released by Global Wind Energy Council, the newly installed wind energy capacity was 54.6 GW, which makes the global cumulative wind energy capacity 486.7 GW (GWEC 2016). Figures 2.1 (a) and (b) show the new installed and cumulative wind energy capacities from 1997 to 2016 respectively.

Figure 2.1 (a) global annual and (b) cumulative installed wind capacities from 1997 to 2016 (GWEC 2016)

The majority of the newly installed wind energy capacity is located in developing countries, such as China, Brazil and India which shows in Figure 2.2. These data demonstrate that the wind energy industry has a very huge market with a promising future. It can therefore be inferred that it is worth investing in wind turbine related studies in order to improve device performance. Any small improvement in turbine performance could bring significant economical benefits due to the huge newly installed wind energy capacity. Therefore, many scientists and institutions spend tremendous time and resources in this field (GWEC 2016). Furthermore, due to the complexity of wind turbine design problems and the constraints of resources, there are still a lot of critical issues that could not be fully solved. The fluid-structure interaction is one of these important issues which will be elaborated in this chapter, because it is closely related to the wind turbine power efficiency, fatigue failure and stability of operation. With the advance of the global wind energy industry, there will be thousands of new wind turbines to be designed and installed all over the world. It is expected that research work in this field would be useful in improving the design of the wind turbine and enhance its efficiency and life span. To carry out the investigation, some basic concepts of wind turbine and aerodynamics will be necessary. In the following sections, the wind turbine

configuration, the aerodynamics of horizontal axis wind turbines and wind turbine rotor design will be reviewed.

Figure 2.2 Percentage of new installed wind capacity in different countries in 2016 (GWEC 2016)

2.1.1.2 Variety of wind energy application and examples

Wind energy has been used to drive the sailing ship since the 7th century. The single-mast sailing ship appeared in the 7th century and the multiple-mast sailing ship was firstly recorded in the 15th century are shown in Figures 2.3 (a) and (b) respectively. The usage of wind energy as thrust power for ship is a significant development in human history. It solved the problems of slow moving speed of the ship as well as relying on manpower to swing the paddle. Long distance shipping became available after the sailing ship showed up and allowed for significant increase in trade and commerce. With the invention of the steam

engine in the 18th century, the sailing ship gradually disappeared from ocean transportation.

Another early application of wind technology, the wind mill was invented by the Persians in the 6th century and was employed for irrigation and grain milling. The Holland and American wind mills are shown in Figures 2.3 (c) and (d), respectively. The energy generated by wind mills played an important part in Europe before and during the period of the Industrial Revolution (Manwell et al. 2002). In Holland, for example, there were all together more than 12,000 windmills working in the 18th century being used not only to grind grain and squeeze oil, but also to drain water from wetlands since more than two thirds of the country lands were below sea level. Although the windmill became less important with the development of the combustion engine in the 20th century, the merits of wind energy such as manpower free and sustainable, makes the windmill still very popular in Holland today.

Wind energy can also be used to provide home heating energy requirements. The wind turbine can generate electricity which subsequently converted into heat. The second way is to use the wind turbine to drive a compressor engine to condense air under adiabatic condition. Heat is then released. The third is to employ the wind turbine to drive blenders to mix liquids inside a container to generate heat.

(a)

(b)

(c)

(d)

Figure 2.3 (a) single-mast and (b) multiple-mast sailing ship (c) Holland wind mill and (d) American wind mill

2.1.2 Wind turbine configuration

Research into wind turbine for electricity generation started at the beginning of the 19th

century. Charles F. Brush was the pioneer of wind electricity research and installed the first automatic wind turbine to generate electricity in Ohio (USA) (Manwell et al. 2002). This wind turbine was rated at 12kW with a horizontal axis and had 144 blades made from wood. Poul la Cour from Denmark made a huge contribution in the development of modern high speed horizontal axis wind turbines by discovering that a turbine with fewer blades and a high rotating speed was more efficient. Additionally, the vertical axis wind turbine (VAWT) appeared around 1930-1940 due to an invention by a Finish engineer Savonius. However, this kind of wind turbine which consisted of several S-shaped blades had the disadvantage of low power efficiency, as compared to HAWT. This resulted in lower competitiveness in the global energy market. Another kind of VAWT is Darrieus type which is driven by lift forces generated from the blade (Manwell et al. 2002). According to the shape of the blade the Darrieus could be classified into several categories as shown in Figure 2.4. Furthermore, Darrieus VAWT has the advantage of high speed, simple structure and low building cost. Disadvantages included small starting torque and relatively low power efficiency.

Figure 2.4 Different types of Darrieus vertical axis wind turbines

Both Savonius and Darrieus VAWTs could not satisfy the requirements of high power efficiency and gradually disappeared from the global market. From the 1980s the modern high speed three-blade HAWT became the mainstream of the market and replaced all other kinds of wind turbines. The capacity of the wind turbine has kept on increasing and those

with a capacity around 1 MW are the most common in the market (Manwell et al. 2002). The development of the wind turbine demonstrates the trend moves towards high speed single-machine capacity. Wind farms with hundreds of wind turbines installed have appeared, many of them being offshore (Goosen et al. 2014).

One example of vertical-axis wind turbine is shown in Figure 2.5 (a) and the other is the horizontal-axis wind turbine as shown in Figure 2.5 (b).

Figure 2.5 (a) vertical axis wind turbine and (b) horizontal axis wind turbine (Manwell et al. 2002)

When operating the horizontal-axis, wind turbine has to be asserted and installed to face the direction of the wind while the vertical-axis wind turbine is independent of the wind direction. When the wind turbine is working under a constant wind speed, the lift force on the HAWT rotor is almost constant while the lift force on VAWT rotor varies periodically. This merit makes HAWT works more efficiently than VAWT (Vries 1983). According to the statistics (GWEC 2015), most of the wind turbines which have been installed all around the world are HAWT. Taking into account of all these factors, HAWT was chosen for the present work. Within the category of HAWT, there are mainly two kinds of different HAWT based

on the rotor configuration - upwind and downwind of the tower which are shown in Figures 2.6 (a) and (b) respectively.

Figure 2.6 Two kinds of rotor configurations: (a) upwind and (b) downwind

Downwind configuration allows the blades to deflect from the tower when the thrust loading increases, but for upwind turbine the yaw is supposed to be controlled. The downwind wind turbine will generate a lot of noise when the wind passes by the tower and generate addition moments which will cause yaw loads when the rotor changes in wind direction. Based on these reasons, the rotor configuration will be selected as upwind in the present study. According to the standard of small size wind turbine defined by IEC (2013), the small wind turbine should have a rated power of less than 50kW.

Another important parameter needs to be decided is the tip speed ratio. The tip speed ratio λ , is the ratio of the circumferential velocity of the blade tips to the wind speed, is shown in equation (2.1)

$$\lambda = \omega R / V_1 \quad (2.1)$$

Where ω is the rotor angular velocity, R is the radius of rotor and V_1 is the incoming wind speed. A typical performance curve for a high speed wind turbine is shown in Figure 2.7. This curve is the variation of power coefficients, $C_p = P / (0.5 \rho S V_1^3)$ which will be derived and presented as equation 2.14 in page 36, note that P is the power, ρ is the density and S is the area of rotor, versus tip speed ratio, λ (Burton et al. 2011). The curve demonstrates that the maximum achievable power coefficient will appear when the tip speed ratio is around 6, which is selected in the present work.

Since the rated power of the selected wind turbine is fixed at 5kW and is classified as a small wind turbine, it is clear from Figure 2.8 that the three-blade rotor has a higher maximum achievable power coefficient than two-blade rotor (Burton et al. 2011). Although the maximum achievable power coefficient will go up as the number of blades increases under the same tip speed ratio (Wilson et al. 1976), the three-blade rotor has a simple structure and yields the lowest system cost (Hansen and Butterfield 1993), the rotor of the present work will be configured as three blades.

Figure 2.7 Power coefficient versus tip speed ratio curve (Burton et al. 2011)

Figure 2.8 Maximum achievable power coefficients as a function of number of blades (Burton et al. 2011)

2.2 Wind turbine rotor design

2.2.1 Aerodynamics of airfoil

Since the HAWT has to be orientated such that its horizontal axis is parallel to the wind velocity, which is along the stream wise direction (i.e. X axis), and also wind turbine blades are long and slim in structures, hence the span wise velocity component is much lower than the streamwise component. Therefore, it is reasonable to assume that flow at a small interval length of blade radial position is two dimensional. Figure 2.9 shows a two dimensional flow passes an airfoil element. The force F , acting on this element of airfoil, can be resolved into two directions, one is the lift force, L , which is perpendicular to the wind velocity, V_1 , and the other is the drag force, D , which is parallel to V_1 , are demonstrated in Figure 2.9. For the case of HAWT, the lift force is the main component in rotating the wind turbine blade and thus generating the electricity and the drag force does not contribute to the output. Therefore, the main objective is to maximize lift force to drag force ratio. A chord, c , is defined as the length of a straight line from the leading edge to the trailing edge of an airfoil as shown in Figure 2.9.

Figure 2.9 Definition of lift, drag, angle of attack, chord and aerodynamic moment

Note that the origin point is located at the aerodynamic center

Angle of attack, α , is defined as the angle between the chord and the direction of incoming wind velocity as shown in Figure 2.9. Aerodynamic moment, M , is defined as the product of resultant force, F , and the length to aerodynamic center which is usually located on the chord at $c/4$ position from leading edge. Lift and drag coefficients C_l , and C_d , are defined respectively as follow:

$$C_l = L / 0.5 \rho V_1^2 c \quad (2.2)$$

$$C_d = D / 0.5 \rho V_1^2 c \quad (2.3)$$

Where ρ is the density of air.

It is known that when the flow passes by a curvature surface, a pressure gradient will be induced. This pressure gradient will cause a centripetal force in radial direction. The fluid

flow over the upper surface will have a trend to leave the surface which will result in low pressure and the fluid flow over the lower flatter surface will have a trend to push the surface which will lead to high pressure. This pressure difference gives a lifting force on the airfoil (White 1991). The lift force is the driving force for the wind turbine to rotate and should be enhanced.

The drag force is due to the boundary layer separates from the upper surface of the airfoil and generates a lot of viscous vortex bubbles. These vortex bubbles will consume some percentages of the total wind energy and do not contribute to the electricity power generation. The drag force has not only reduced the power generation but also induced flow instability which is harmful to the wind turbine operation and should be suppressed (White 1991).

The coefficients C_l and C_d are functions of the angle of attack, α , the Mach number, Ma , which represents the ratio between V_1 and the speed of sound and the Reynolds number, Re . The incoming wind speed is around 10m/s which equals to the Mach number of 0.03. According to Houghton and Carpenter (2003) the effect of Mach number can be ignored. Note that Re is defined as:

$$Re = cV_1/\nu \quad (2.4)$$

where ν is the kinematic viscosity.

The lift coefficient C_l increases linearly with α to reach a maximum value, and it can maintain at the maximum for a couple of degrees in α and then decrease dramatically, which means that the airfoil has reached stall status. These are depended on the Reynolds number. For the drag coefficient, C_d , which will keep increasing for the further increase of Reynolds number after the airfoil steps into the stall status. Figures 2.10 (a) and (b) show the curves of C_l versus C_d of airfoil S822 (Selig and Granahan 2004) and S823 (Selig et al. 1995) which

will be employed in this study. It can be observed that when selecting the normal working angle of attack for each element of the blade, not only C_l should be considered, but also should include C_d . It is necessary to select the angle of attack at the point where the C_l/C_d ratio is the maximum to ensure that each element of blade will achieve the best performance. For the airfoils of S822 and S823 (Appendix A), it is clear that the drag coefficient increases slower than lift coefficient from the point of maximum C_l/C_d ratio to the point where C_l reaches maximum value. According to Burton et al. (2011) the drag effect due to the skin friction could be ignored as long as the flow remains attach to the blade because the losses caused by drag is only critical when the wind turbine is operating at high incoming wind speed. Wind turbine power coefficient, C_p , is defined as the ratio of the power generated by wind turbine to the total wind power in equation (2.14), please refer to page 36. The variation of maximum power coefficient versus tip speed ratio is shown in Figure 2.11.

Figure 2.10 C_l vs C_d curve of airfoil (a) S822 (Selig and Granahan 2004) and (b) S823 (Selig et al. 1995) under different Reynolds number

Figure 2.11 Variation of maximum power coefficient for different lift to drag ratios

Since the lift to drag ratios for both S823 and S822 are relatively low as well as the design tip speed ratio, the angle of attack will be selected near around the point where C_l achieves the maximum value.

The stall status of airfoil is not only related to α but also depends on Reynolds number. Schlichting (1968) showed that the Reynolds number dependency is related to the point on airfoil, where the boundary layer transits from laminar flow to turbulent flow as shown in Figure 2.12.

Figure 2.12 Transition of flow from laminar to turbulent along upper surface of airfoil

The way of an airfoil stall is normally decided by its shape geometry. The thin airfoil which has sharp nose is much more easier encountering sudden stall than the thick airfoil which has a low curvature at leading edge (Crimi and Reeves 1976). This is because different kind of airfoil will lead to different process of boundary layer separation which occurs at the upper surface of airfoil. If the flow separation begins near around the trailing edge of the airfoil and keep increasing slowly with the augment of the angle of attack, then a so called soft stall phenomenon will happen. On the other hand, if the boundary layer separation occurs near around the leading edge of the airfoil, then the major portions of the boundary layer will separate from the upper surface and result in a huge loss of lift force (Boiko et al. 2015). The separation of viscous boundary layer is a very complicated physical phenomenon and related to a lot of factors including the type of airfoil, the Reynolds number, the roughness of surface and the flow speed (Cebeci and Smith 1974).

The flow field around the airfoil can be divided into two parts by the stagnation streamline as shown in Figure 2.12 since the air flows along both the upper and the lower surfaces. It is known that the flow velocity is zero at the stagnation point and the boundary layer thickness become very small. When the flow passes over the airfoil leading edge, it will be accelerated because the leading edge is close to the stagnation point (Dovgal et al. 1994). Since the Bernoulli equation can be applied outside the boundary layer and considering that the flow

has been accelerated near around the leading edge, then it can be shown that the pressure of the flow will drop as shown in Figure 2.13 (a) and can be expressed as $\partial p / \partial x_b < 0$, where p is the flow pressure and x_b is the distance from the leading edge stagnation point. The flow pressure gradient on the lower surface is much smaller than that on the upper surface due to relatively small curvature. Since the pressure should be the same due to the Kutta condition (Crighton 1985) at the trailing edge, then it can be deduced that the pressure must increase which will lead to a positive pressure gradient $\partial p / \partial x_b > 0$. The adverse pressure gradient will result in boundary layer separation which can be shown by applying the Navier-Stokes equations (Appendix B) at the airfoil surface and set the velocity as zero using no slip condition. Then the whole Navier-Stokes equations can be simplified and the profile of velocity u can be determined as shown in Figure 2.13 (b).

Figure 2.13 Profile of airfoil boundary layer (a) favourable pressure gradient and (b) adverse pressure gradient

From Figure 2.13 (a) it can be deduced that with a favourable pressure gradient the boundary layer will keep attaching over the whole airfoil surface. From Figure 2.13 (b) it can be observed that as the distance from the leading edge stagnation point becomes larger, the u

velocity profile with a positive pressure gradient turns into an inverted S-shape and flow separation will prone to happen. Since the boundary layer separation is closely related to the dramatic increased drag force, then it is very crucial to keep a favourable pressure gradient as long as possible and keep the flow attached to the airfoil surface (White 1991). Although the boundary layer will keep in laminar condition for small values of x_b , beyond some distances the laminar boundary layer will begin to convert to turbulent flow. Therefore, it is better to keep the boundary layer in laminar flow condition for a long distance from the leading edge and also try to keep the favourable pressure as long as possible.

As the stall phenomenon is closely related to boundary layer separation, it implies that the main target in blade design is to reduce drag and trying to make the flow attach to the blade surface as long as possible. In other words, it has to ensure that the airfoil is working below the angle of attack in which C_l/C_d ratio is the maximum (Bertin and Smith 1989).

To ensure the airfoil elements which constitute the wind turbine blade is not stepping into the stall situation, considering a small element dr from a rotating wind turbine rotor as shown in Figure 2.14 is used to perform the flow analysis and the velocity triangle is demonstrated in Figure 2.15. Note that in this figure, V_1 is wind velocity (which flows perpendicularly into the paper in Figure 2.14), V_{rot} is the blade rotating velocity, V_{rel} is relative wind velocity which is the vector difference between V_1 and V_{rot} , α is angle of attack, θ is twist angle and ϕ is relative wind velocity angle. The relationship among these three angles is as follow:

$$\alpha + \theta = \phi \quad (2.5)$$

To simplify the illustration assumes that there are no tip and root losses. The wind speed and angular velocity of the rotor is fixed, then it can be expected that the relative wind velocity angle ϕ , of each element of the blade will increase from tip to root. To maintain a constant angle of attack α , the twist angle θ has to increase along the blade radial direction from tip to

root based on the relationship of equation (2.5). This is the main reason why most of the HAWT blades are twisted. Tong (2010) illustrates that the tip and body portions of turbine blade contribute to the most part of power generation and the root zone only contributes a small percentage. In wind turbine blade design, it is paramount to ensure that the tip and more blade body percentage of the wind turbine to generate work output around the maximum angle of attack, however, as the twist angle has to increase from the tip to root, it is unavoidable that the performance of blade elements near the root has to be sacrificed because when the tip portion is working under the overrated wind speed the root portion has stepped into the stall condition earlier than the tip part.

Although it is necessary that theoretically the angle of attack at the blade tip should be selected at the maximum, in practice the designed angle of attack will be given a little buffer from the maximum value, i.e. smaller than the maximum angle of attack. This is to avoid the wind turbine blade working under the stall condition when encountering the gust which may result in the unfavorable angle of attack due to the large twisting angle at the blade root (Zhao and Xu 2011).

Figure 2.14 Rotor of wind turbine

Figure 2.15 Velocity analysis of section dr

2.2.2 Blade design theory

There are a lot of wind turbine blade design methods and among these methods the Wilson method is most widely employed. This design method has included the Betz theory, wake theory, blade element theory and the momentum theory (Wilson and Lissaman 1974).

2.2.2.1 Betz theory

The theory established by A. Betz in 1919 is the first to describe the maximum amount of energy that a turbine can extract from the wind. The important assumptions of this theory are:

- (1) the ideal rotor has infinite number of blades
- (2) the flow can be simplified to an annular stream tube as shown in Figure 2.16
- (3) the flow is stationary, incompressible and frictionless
- (4) no rotation in the wake
- (5) the force applied on rotor is distributed evenly

Considering an annular stream tube contains a wind turbine as shown in Figure 2.16

Figure 2.16 Energy extracting actuator and annular stream tube

Note that V is the wind speed after rotor, V_2 is the far downstream wind speed, S_1 is the area of actuator disk before rotor, S is the area of rotor and S_2 is the area of actuator disk after rotor. From the mass conservation equation it can be shown that:

$$S_1 V_1 = S_2 V_2 = S V \quad (2.6)$$

The force acts on wind turbine blade, F , can be derived as:

$$F = \rho S V (V_1 - V_2) \quad (2.7)$$

Then the power of the wind turbine rotor, P , can be expressed as:

$$P = FV = \rho S V^2 (V_1 - V_2) \quad (2.8)$$

The axial momentum equation is applied within the circular control volume and combined with the mass conservation law, and it can be expressed as following:

Firstly, V can be taken as an average of V_1 and V_2 .

$$V = (1/2) (V_1 + V_2) \quad (2.9)$$

Substitute equation (2.9) into (2.8) the power equation becomes:

$$P = FV = (1/4) \rho S (V_1 + V_2) (V_1^2 - V_2^2) \quad (2.10)$$

By taking the derivative of equation (2.10) with respect to V_2 because V_1 , ρ and S are constants, hence the power is only the function of V_2 :

$$dP/dV_2 = (1/4) \rho S (V_1^2 - 2V_1 V_2 - 3V_2^2) \quad (2.11)$$

To obtain the maximum power, set equation (2.11) to 0 and lead to the result:

$$V_2 = (1/3) V_1 \quad (2.12)$$

Substitute equation (2.9) and (2.12) into (2.10) then the maximum power of an ideal wind turbine rotor is as follow:

$$P_{max} = (8/27) \rho S V_1^3 \quad (2.13)$$

Since the power coefficient is defined as follow:

$$C_p = P / (0.5 \rho S V_1^3) \quad (2.14)$$

Therefore, the maximum power coefficient $C_{p_{max}} = P_{max} / (0.5 \rho S V_1^3) = 16/27 \approx 0.593$ which is named as the Betz limit. This conclusion demonstrates that under an ideal circumstance the wind turbine power coefficient cannot be higher than 0.593.

2.2.2.2 Rotor Wake Theory

The rotor wake theory takes the flow rotation induced by the rotor into account and assumes that:

- (1) the effect of airfoil drag and blade tip loss were ignored
- (2) the periodic effect of the flow due to the limited blade number was ignored
- (3) the blade element in radial direction is independent from each other

Figure 2.17 Different kind of vortex behind wind turbine

The induced wind speed can be treated as the combination of three kinds of vortex as shown in Figure 2.17:

- (1) center vortex which is located behind the rotating shaft
- (2) attached vortex which is located at the blade main body
- (3) helical vortex which is located at the blade tip

Due to the presence of these vortices, the axial and tangential direction of the wind velocity will change. The axial (X direction) and rotational (rotating around X axis) direction induction factors are denoted as a and b respectively, and will be introduced to represent the effect of the rotor to the flow. From the Pramod (2011) it can be known that at the rotating plane, the axial wind velocity is:

$$V = (1 - a)V_1 \quad (2.15)$$

Where a is introduced into the equation to represent the obstruction effect of wind turbine rotor to the incoming wind speed.

For the rotational direction due to the vortex motion, there will be a tangential rotating angular velocity in downstream and zero angular velocity in upstream. When the blade rotates at a constant angular velocity ω , then based on the Rotor Wake Theory (Burton et al. 2011) the flow which passing through the rotor will be induced to rotate at an angular velocity Ω in the same direction with the wind turbine rotor, where Ω represents the induced angular velocity. The expression of induced tangential velocity is:

$$V_t = b\omega r \quad (2.16)$$

Based on the Betz theory (Betz 1966) the following equation of flow angular velocity to rotor at the location of rotor plane can be derived:

$$\omega + \Omega/2 = (1 + b)\omega \quad (2.17)$$

Then the rotational induced factor will be:

$$b = \Omega/(2\omega) \quad (2.18)$$

Where b is rotational induction factor. Taking the induced tangential velocity into account, the tangential velocity of rotor at the location of radius r is:

$$U_T = V_t + \omega r = b\omega r + \omega r = (1 + b)\omega r \quad (2.19)$$

2.2.2.3 Blade Element Theory

In Blade Element Theory the blade is divided into several small blade elements in radial direction. Each blade element can be treated as two dimensional airfoil and assume that there is no interaction between each element. The integration is carried out for the force and torque

act on each element along the radial direction as shown in Figure 2.18, then the total force and torque applied on the whole blade can be obtained (Manwell et al. 2002).

Figure 2.18 Local force and velocity on a wind turbine blade element at radius r

$$dL = (1/2) \rho V_{rel}^2 c C_l dr \quad (2.20)$$

$$dD = (1/2) \rho V_{rel}^2 c C_d dr \quad (2.21)$$

$$V_{rel} = V_1(1-a)/\sin \phi \quad (2.22)$$

$$dF_x = (1/2) \rho V_{rel}^2 c dr (C_l \cos \phi + C_d \sin \phi) \quad (2.23)$$

$$dF_y = (1/2) \rho V_{rel}^2 c dr (C_l \sin \phi - C_d \cos \phi) \quad (2.24)$$

The expressions of thrust and torque on the blade element at the location of radius r are:

$$dT = (1/2) \rho V_{rel}^2 B c dr (C_l \cos \phi + C_d \sin \phi) \quad (2.25)$$

$$dM = (1/2) \rho V_{rel}^2 B c r dr (C_l \sin \phi - C_d \cos \phi) \quad (2.26)$$

Where T is the thrust on the blade, M is the torque on the blade and B is the number of the blade of the whole rotor.

2.2.2.4 Blade Momentum Theory

Wind turbine momentum was first suggested by William Rankine to describe the relation between the force acts on wind turbine and the incoming wind speed. As shown in Figure 2.14 the thrust and moment applied on the annular tube zone $(r, r + dr)$ of wind turbine blade is (Spera 1994):

$$dT = 4\pi\rho r V_1^2 (1 - a) a dr \quad (2.27)$$

$$dM = 4\pi\rho r^3 V_1 \omega (1 - a) b dr \quad (2.28)$$

Equating equations (2.25) and (2.27), the following equation could be obtained:

$$a/(1 - a) = Bc(C_l \cos \phi + C_d \sin \phi)/(8\pi r (\sin \phi)^2) \quad (2.29)$$

Equating equations (2.26) and (2.28), the following equation could be obtained:

$$b/(1 + b) = \omega Bc(C_l \sin \phi - C_d \cos \phi)/(8\pi r (\sin \phi \cos \phi)) \quad (2.30)$$

From the velocity triangle shown in Figure 2.18, the following equation can be obtained:

$$\tan \phi = (1 - a)/((1 + b)\lambda_r) \quad (2.31)$$

Where λ_r is the local tip speed ratio and is defined as follow:

$$\lambda_r = \omega r / V_1 \quad (2.32)$$

Applying the Bernoulli's equation before and after the rotor in Figure 2.16 the change of pressure between these two locations can be expressed (Spera 1994):

$$P' = \rho r^2 (\omega + \Omega/2) \Omega \quad (2.33)$$

With P' obtained from equation (2.33), the thrust dT can also be written as:

$$dT = P' dS = 2\pi \rho r^3 (\omega + \Omega/2) \Omega dr \quad (2.34)$$

From equations (2.17), (2.18), (2.27), (2.32) and (2.34), the following expression can be obtained:

$$b(1 + b)\lambda_r^2 = a(1 - a) \quad (2.35)$$

From equation (2.28) the power, dP_{ele} , of a blade element dr at location r can be obtained as:

$$dP_{ele} = \omega dM = 4\pi \rho r^3 V_1 \omega^2 (1 - a) b dr \quad (2.36)$$

Therefore, the expression of power coefficient of the blade element is:

$$dC_p = (8b(1 - a)\lambda_r^3/\lambda^2) d\lambda_r \quad (2.37)$$

The power coefficient in equation (2.37) is derived, based on the assumption of an infinite number of blades. For a rotor with a finite number of blades the vortex distribution in the wake is different from that of a rotor with infinite number of blades. Therefore, the loss at the location of tip and hub of the blade could not be ignored. Prandtl derived a total loss factor F_T to include their effects (Hansen 2008).

$$F_T = F_{tip} F_{hub} \quad (2.38)$$

$$F_{tip} = (2 \cos^{-1} e^{-(B(R-r)/(2R \sin \phi))})/\pi \quad (2.39)$$

$$F_{hub} = (2 \cos^{-1} e^{-(B(r-r_{hub})/(2r_{hub} \sin \phi))})/\pi \quad (2.40)$$

Where F_{tip} (see equation 2.39) and F_{hub} (see equation 2.40) are the tip loss factor and the hub loss factor respectively. Note that r_{hub} is the radius of hub.

By considering the total correction factor and according to Glauert (1935) the equations (2.4), (2.35) and (2.37) will be transformed to equations (2.41), (2.42) and (2.43) respectively.

$$Re = V_1(1 - a)c/(v \sin \phi) \quad (2.41)$$

$$b(1 + b)\lambda_r^2 = a(1 - aF_T) \quad (2.42)$$

$$dC_p = (8F_T b(1 - a)\lambda_r^3/\lambda^2)d\lambda_r \quad (2.43)$$

The core mission in the designing process of a wind turbine blade is to get the maximum value of dC_p in equation (2.43) under the constrains of equations (2.31), (2.38), (2.39), (2.40) and (2.42) based on the given conditions. The iteration method is applied to obtain chord and twist angle (Ingram 2005). Based on the BEM method, the wind turbine blade could be designed with relative high power coefficient which ensure the wind energy play an important role in the renewable energy industry. The details of the design procedure are illustrated in Figure 2.19.

Figure 2.19 Flow chart of wind turbine blade design process

2.3 Fluid-structure interaction of wind turbine

2.3.1 Introduction

The fluid-structure interaction is a very important phenomenon which is closely related to the power efficiency, life span and stability during the wind turbine operation. To have a better understanding on how the flow will cause the blade deformation and vice versa, becomes a critical issue for researchers. A literature review of fluid-structure interaction of wind turbine is presented in this section.

2.3.2 Basic theory of fluid-structure interaction

2.3.2.1 Theory description

Fluid-structure interaction problem can be described by a set of highly non-linear equations. When solving the fluid-structure interaction problem, the instantaneous solution of the coupling equation should be obtained. These solutions should fulfill the boundary conditions on the interface of the fluid to structure (Bathe et al. 1999). One of the challenges is the method in describing the dynamics of fluid and structure is different from each other, because the fluid is using Euler method and the structure is described by Lagrange method. The Navier-Stokes equation is described using Eulerian method and using a fixed reference frame to derive all the parameters such as velocity and pressure (Galdi and Rannacher 2010). In fluid-structure interaction, the fixed location is not stationary but moving with time, so the Navier-Stokes equation cannot be directly applied.

To adapt to the characteristics of the flow domain which is changing with the time, the most common way is to combine the Eulerian and Lagrangian methods. Combining these two methods to solve the FSI problems will encounter some difficulties. One problem is that the flow domain is time-dependent and relies on the deformation received from the structure domain. Another issue is that the structure domain needs all the parameters such as velocity and stress from the fluid boundary. A common method adopted to solve this problem is to separate the flow and the structure domains. Each domain will employ an iterative method to obtain a converged result which will satisfy both domains and interface conditions. The Arbitrary Lagrangian-Eulerian (ALE) method is thus developed based on solving the separated domain and can simultaneously deal with the implicit coupling FSI problems. It is also a widely used method to solve the moving boundary conditions due to the inconsistent

coordinates in fluid-structure interaction problem (Donea et al. 1982). Figure 2.20 demonstrates the FSI calculation process chart.

Figure 2.20 FSI calculation process chart

From Figure 2.20 it can be seen that the FSI calculation process starts from the flow domain and will transfer the result to structure domain within one iterative time period. All the data transmission between these two domains will need the interface as a media. Therefore, to precisely capture the movement of the interface is very critical for accurately solving the FSI problems (Legay et al. 2004).

2.3.2.2 Solving method

According to different solving approaches, the FSI solving method can be divided into two main categories: the direct approach and the iterative approach. The direct approach will establish the fluid and structure equations and solve them together. Although the accuracy of

the direct approach is very high, the computational work will be very tedious because it demands to solve a lot of nonlinear equations at the same time. Due to the different physical and mathematical characteristics of flow and structure domains, the complexity and the calculation loads will exponentially increase and bring a lot of difficulties for the computational process (Rugonyi and Bathe 2001). In iterative approach the equations of the fluid and structure will be established and solved separately and the solution parameters will be passed from one domain to the other domain iteratively until the convergence of computation is reached. The iterative approach reduces the complexity of the computational process, simplifies the treatment of the implicit equations and is easily used to establish the flow and structure domain model. Therefore, the iterative approach becomes the mainstream method for solving the FSI problems.

Based on the involvement of the stagger iteration within one time step, the iterative approach can be divided into two different types: the strong coupling and the weak coupling. The strong coupling can reduce the time delay and increase the time accuracy in FSI computational response by using multiple stagger iteration between the flow and the structure systems. The weak coupling method employs single stagger iteration way which makes the computational load equals to normal unsteady CFD analysis. Although the accuracy of weak coupling method is only first order, it provides a way to treat the flow domain and the structure domain separately. The format of weak coupling is relatively easy and does not require the total consistency of the mesh on the interface. All of these advantages make the weak coupling method become more suitable for solving the practical problems (Galdi and Rannacher 2010).

2.3.2.3 Interface data transmission

There are two types of arbitrarily shaped interface tracking techniques: the interface tracking technique and the interface capturing technique. The main difference between these two techniques is that for the FSI interface computation the interface tracking technique is based on the Lagrange method and the interface capturing technique is based on the Euler method (Takagi et al. 2012). The significant characteristics of the two techniques are shown in Figure 2.21.

Figure 2.21 Comparison of two kinds FSI interface dealing techniques (a) interface tracking technique and (b) interface capturing technique

Figure 2.21 (a) shows that the interface tracking technique uses the moving boundary conforming grid to follow the movement in structure domain and the deformation in flow domain. This technique uses the movement of grid to conform the change of the grid shape in order to follow the FSI interface which ensures the computational accuracy near around the critical area of moving boundary line. Since the movement of grid keeps consistently with the interface then this method can satisfy the moving equation automatically. Due to the shape change of structure the mesh needs to keep updating in partial mesh zone or even the whole mesh zone, so that the quality of the mesh can be guaranteed. In interface capturing technique which has been illustrated in Figure 2.21 (b), the physical equations of flow and the structure domains are constructed in fixed Euler grid and Lagrange grid respectively (Takizawa and Tezduyar 2011). In the interface tracking technique the boundary conditions could be applied at the desired location near the interface because the interface information can be known at every moment. In the interface capturing technique the boundary will appear in the governing equations which will result in the implicit boundary information (Sarrate et al. 2001).

2.3.3 Governing Equations

The equations of motion for an incompressible Newtonian fluid using the ALE method are given as follows (Rugonyi and Bathe 2001):

Momentum equation:

$$\rho(Dv/Dt) + \rho[(v - \hat{v}) \cdot \nabla]v = \nabla \cdot \tau + f^B \quad (2.44)$$

Mass conservation equation:

$$(Dv/Dt) + \rho(v - \hat{v}) \cdot \nabla + \nabla \cdot v = 0 \quad (2.45)$$

Energy equation:

$$\rho(De/Dt) + \rho(v - \hat{v}) \cdot \nabla e = \tau f^{st} - \nabla \cdot q + q^B \quad (2.46)$$

Where D/Dt is the total time derivative in ALE frame, e is the specific internal energy, v is the velocity vector, $\hat{v}$ is the velocity vector in ALE frame, τ is the fluid stress tensor, f^B is the body force vector, f^{st} is the velocity strain vector where $2f^{st} = \nabla v + (\nabla v)^T$, q is the heat flux tensor, q^B is heat per unit volume, $(\nabla \cdot)$ is divergence operator, ∇ is gradient operator and $(\cdot)$ is dot product. If $\hat{v} = 0$ then it means that the frame is not moving and all the equations become Eulerian form. If $v = \hat{v}$, then it means that the fluid is moving together with the frame then the Lagrangian formulation will be recovered.

The governing equation of the motion of the structure is shown as (Rugonyi and Bathe 2001):

$$\rho D^2 u / Dt^2 = \nabla \cdot \tau + f^B \quad (2.47)$$

Where u is the vector of the structure displacement.

To solve the fluid-structure interaction problem, the equations from (2.44) to (2.47) require to be calculated together based on the determined boundary conditions and stable solution approach.

2.3.4 Flow induced vibration on wind turbine blade

The blade is the component of wind turbine which is subjected to the most complicated force. The aerodynamic loads, the inertial loads and the gravitational loads are the three most important loads acting on a wind turbine blade. The blade is a flexible slender object on which different kinds of mechanical vibrations will be easily induced. The blade is a typical aerodynamic component on which the aerodynamic damping and the lift force will be encountered during the rotation. Under such situation the wind turbine blade becomes an aero-elastic component taking the comprehensive effect of mechanical vibration and aerodynamic damping simultaneously.

The comprehensive effects of the aerodynamic loads, the inertial loads and the gravitational loads will cause three main kinds of turbine blade deformation. Flap is the blade bending movement in the direction which is perpendicular to the rotating plane, swing is the blade bending displacement in the rotating plane and twist is the blade torsion around the aerodynamic center of each blade element in rotational direction (Bazilevs et al 2011). The three kinds of deformation of blade are shown in Figure 2.22.

Figure 2.22 Three main kinds of blade deformation: (a) flap, (b) swing and (c) twist

The interaction among these three kinds of deformation and the aerodynamic loading will cause the aero-elastic problem (Rasmussen et al. 2003). When the deformation of the blade is occurring then the bending will cause the change of the angle of attack, which will cause the lift force variation, and subsequently will change the aerodynamic torque, this will result in a feedback of the bending to the bending-twist coupling vibration. When the circumstance is mature then a positive feedback will be formed which will cause the self-excited oscillation. When the amplitude of self-excited oscillation is increased, normally there will be a severe

damage to the wind turbine blade and this kind of unstable aero-elastic vibration is named as the fluttering phenomenon.

2.3.5 Blade natural frequency and vibration mode

When the excitation frequency of the blade approaching the natural frequency, the resonance will occur. To avoid this situation, within the main operating range the designed blade natural frequency should have a large enough difference from the excitation frequency. Therefore, it is necessary to predetermine the blade natural frequency vibration mode. According to Tong (2010) the behavior of the wind turbine blade in fluid-structure interaction is similar to that of a cantilever beam. Based on the mechanical vibration theory the blade can be treated as a distributed parameter system. Modal analysis will be used to obtain the natural frequency and mode of the blade. As mentioned earlier, there are different orders of the main mode of an elastic object and they are orthogonal to each other. Therefore, the displacement function of the blade $y(x, t)$ is expressed as a product of a space function $\gamma(x)$ and a time function $g(t)$ as shown in equation (2.48)

$$y(x, t) = \gamma(x)g(t) \quad (2.48)$$

where $\gamma(x)$ is the mode of the elastic object which is only related to its particle position, $g(t)$ which is the pattern of the vibration and only depends on time.

Since the wind turbine blade is a slender body which means that length of blade is normally 10 times or even larger than the width, then the thin beam theory or the Euler-Bernoulli beam theory can be applied to the wind turbine blade. According to the Euler-Bernoulli beam theory the rotation of the beam cross section could be ignored as compared to the translation. The angular distortion which is caused by the shear stress can be neglected as compared to the bending deformation (Rao 2007).

The natural frequency and vibration mode of an uniform cantilever beam which is similar to that of blade can then be obtained using equation (2.49) together with the following four boundary conditions (Bathe et al. 1999):

$$d^2(EI(d^2\gamma(x)/dx^2))/dx^2 = m\omega_n^2\gamma(x) \quad (2.49)$$

$$\gamma(0) = 0 \quad (2.50)$$

$$(d\gamma(x)/dx)_{x=0} = 0 \quad (2.51)$$

$$(d^2\gamma(x)/dx^2)_{x=L} = 0 \quad (2.52)$$

$$(d^3\gamma(x)/dx^3)_{x=L} = 0 \quad (2.53)$$

where E is Young's modulus, I is moment of inertia, x is distance from origin, ω_n is natural frequency, m is mass and L is the length of the blade. Using the boundary conditions of cantilever beam as listed from equations (2.50) to (2.53), each order of the natural frequency and normalized mode can be obtained (Leissa and Qatu 2011).

Figure 2.23 (a) shows the configuration and parameter of a cantilever beam. Figures 2.23 (b), (c) and (d) demonstrate the cantilever beam natural frequency and vibration mode of first order, second order and third order respectively. It can be observed from Figures 2.23 (b) to (d) that the natural frequency will keep increasing as the vibration order augmented and the vibration mode curve will possess $n-1$ crossing nodes with x axis for n_{th} order vibration mode, which indicates the relationship between the number of motionless points during the blade vibration and its order of vibration mode (Beards 1996).

Noted that the parameter β satisfies the equation (2.54).

$$(\cos(\beta_n L)) * (\cosh(\beta_n L)) = -1 \quad (2.54)$$

Where $n=1,2,3,\dots$ is the order of the vibration mode.

Figure 2.23 Cantilever beam natural frequency and vibration mode (a) cantilever beam configuration, (b) first order vibration mode and frequency, (c) second order vibration mode and frequency and (d) third order vibration mode and frequency

To sum up, the present work involved strongly coupling distributed-parameter system interaction problem. The analytical solutions of equations - including the fluid domain governing equations, the solid governing equations as well as the vibration equations can only be obtained under some special simplified cases. The aero-elastic phenomenon introduced in the present work requires to study is very complicated, beside that the numerical simulation requires an economical usage of the limited computing resource available to obtain the detailed blade deflection and the whole flow field information.

2.3.6 Aeroelastic stability of wind turbine

Under the aerodynamic force the elastic body will deform and vibrate which will never slow down the variation of aerodynamic forces. This kind of interaction between elastic body and aerodynamic force will result in complicated aeroelastic phenomenon. Figure 2.24 demonstrates the relationship among the aerodynamic, elastic and inertia forces.

Figure 2.24 Triangle of aeroelastic force

It can be seen from the figure that the combination of aerodynamic and elastic forces forms the static aeroelasticity problem, the combination of elastic and inertia forces forms vibration problem and the combination of inertia and aerodynamic forces forms rigid body aerodynamic problem. Aeroelasticity which is to be considered as the combination of elastic, aerodynamic and inertia forces is complicated in wind turbine field. The strong coupling of these three forces makes aeroelasticity plays an important role in wind turbine operation. The aerodynamic force relies on the relative wind speed passes through wind turbine blade surface. The deforming of the structure will change the shape of the blade due to elastic bending force. The shape change of the blade will affect the angle of attack and then alter the aerodynamic force. Beside that the speed of blade deforming will also lead to the relative wind speed passes by the blade surface, which also changes the aerodynamic force. In turn the aerodynamic force will continuously affect the blade deformation and the acceleration of the wind turbine blade makes this phenomenon a two-way interaction process. Inertia force

also plays an important part in the correlation of aerodynamic and elastic forces which result in the acceleration of the wind turbine blade (Hansen 2007).

There are two main kinds of aeroelastic instability which are most related to wind turbine operation, one is classical flutter and the other is stall flutter. Flutter is a phenomenon when blade is excited to oscillate under the mutual interaction of aerodynamic, elastic and inertia forces. The classical flutter is a combination of the bending vibration and torsional vibration which is actually unavoidable due to twisting of the wind turbine blade, and results in the change of angle of attack as well as the lift force. Therefore, the classical flutter is very harmful to wind turbine blade and can lead to total destruction within a short period of time.

Figures 2.25 (a) and (b) show a typical airfoil section selected from the wind turbine blade in order to illustrate the process of classical flutter in 0° and 90° phases respectively. When the structural damping is taking into account, the total energy of the typical section during the oscillating will gradually decrease due to the energy loss caused by the structural damping and then the whole system is stable. If the energy of the airfoil received during one oscillating period is bigger than the energy loss caused by structural damping, then the energy will be accumulated during the vibration process and lead to infinite vibration response which will trigger the aeroelastic instability.

As shown in Figure 2.25(a), the typical airfoil section bending vibration phase is the same as the torsional vibration within one oscillating period. For simplicity, only the lift generated by angle of attack is being considered. From the analysis it could be shown that within one vibration period, the total energy in which aerodynamic force transfers to airfoil section is zero. Therefore, the vibration response will not be diverged and no aeroelastic instability will occur. The situation shown in Figure 2.25(b) is different from Figure 2.25(a). In Figure 2.25(b) the phase angle of bending vibration has 90 degree out of phase from torsional

vibration. Under such condition, the aerodynamic force always transfers positive energy to airfoil section within one vibration period. If the total positive energy received by the airfoil section within one vibration period is bigger than the energy loss caused by structural damping, then the energy obtained by the airfoil section will be accumulated as the vibration continues. Finally the cumulated energy will lead to aeroelastic instability and then the wind turbine flutter will occur. Normally the occurrence of classical flutter will require the system to have at least two degrees of freedom which are coupled with each other under some specific vibration phase difference (Zhang and Huang 2011).

Another kind of flutter is stall induced flutter which is characterised by a periodic change between the flow separation and normal flow on wind turbine blade surface when approaching the crucial angle of attack. The stall flutter could be very harmful to wind turbine rotor when it is operating near aerodynamic stall condition with high wind speed (Hau 2013). When the wind turbine blades step into stall condition, there will be an increasing of negative angle of attack gradient which leads to negative aerodynamic damping. The negative aerodynamic damping will lead the wind turbine blade into resonance condition in both flapwise and chordwise directions. Aerodynamic damping is a speed related factor which generates from the change of angle of attack and acting in an opposite direction to the wind turbine movement. It is decided by the structure deformation speed. The flapwise aerodynamic damping is much bigger than that of chordwise. Although the aerodynamic damping is present during the blade vibration process, the flutter still can occur under certain condition and lead to the damage of wind turbine blade.

Figure 2.25 Energy obtained in airfoil section within one vibration period (a) zero degree phase difference and (b) 90 degree phase difference

The aeroelastic system of the airfoil section shown in Figure 2.25 can also be considered as the coupling of the structure dynamic subsystem and the aerodynamic subsystem. According to Ding (2011) The structure dynamic subsystem can be expressed as follow:

$$M\ddot{q} + C\dot{q} + Kq = f \quad (2.55)$$

Where q is the generalized coordinate, M , C and K is the mass matrix, damping matrix and stiffness matrix respectively, f is generalized aerodynamics force which is related to the motion of the airfoil. Performing Laplace transformation on equation (2.55), it can be rewritten as following.

$$(s^2M + sC + K)q(s) = f(s) \quad (2.56)$$

Putting $q(s)$ as subject, the equation (2.57) can be obtained:

$$q(s) = E(s)f(s) \quad (2.57)$$

$$\text{Where } E(s) = (s^2M + sC + k)^{-1} \quad (2.58)$$

According to equation (2.57) the structure dynamic subsystem can be treated as a linear system with input f , output q and transfer function $E(s)$ which is demonstrated in Figure 2.26 (a).

Figure 2.26 Aeroelastic system flow chart (a) structure dynamic subsystem, (b) aerodynamic subsystem and (c) feedback coupling of two subsystems

For the aerodynamic subsystem, according to the Grossman aerodynamic theory (Fung 1993) the equations can be expressed as:

$$f = \rho V A_c \dot{q} + \rho V^2 A_k q \quad (2.59)$$

Where A_c and A_k are the airfoil velocity related and displacement related parameter matrix respectively.

Conducting Laplace transformation on equation (2.59), the following equations can be obtained:

$$f(s) = \rho V^2 A(s) q(s) \quad (2.60)$$

$$\text{Where } A(s) = (s/V)A_c + A_k \quad (2.61)$$

According to equation (2.60) the aerodynamic subsystem can be treated as a linear system with input q , output f and transfer function $A(s)$ as shown in Figure 2.26 (b).

The aeroelastic system can be expressed by coupling the equations (2.55) and (2.59). Actually the aeroelastic system is a feedback system which is consisted of the structure dynamic subsystem and the aerodynamic subsystem as demonstrated in Figure 2.26 (c). Based on the modern control theory a feedback system will encounter the dynamic instable problem (Franklin et al. 2009). In Figure 2.26 (c), the ρV^2 part can be treated as a feedback gain which is very crucial to the stability of the airfoil feedback system. When ρV^2 is equal to zero, it means that the system is an open loop system and there is no coupling between the structure subsystem and aerodynamic subsystem. When ρV^2 is within some specified ranges, the system will remain stable. When ρV^2 is over a critical value, then the feedback system will lose its stability. Therefore, from the feedback control point of view the fluttering of airfoil can be treated as a dynamic instability is larger than a specific value ρV^2 to the coupling of structure subsystem and aerodynamic subsystem (Tewari 2015).

The wind turbine blade resonance should be distinguished from the aeroelastic instability. Wind turbine blade will under resonance conditions when the external excitation frequency is getting closed to the blade natural frequency and then the energy can be continuously add to the blade due to the excitation. Aeroelastic instability is caused by the aerodynamic force which adds energy into system but not due to the resonance of natural and excitation

frequencies. Therefore, by only avoiding the blade natural frequency coincidence with the external excitation frequency is not enough to ensure the aeroelastic stability (Hansen 2007).

Figures 2.27 (a) and (b) demonstrate the present 5kW wind turbine blade under different rotating speed (in rpm) and the potential resonance points. Figure 2.27 (a) shows that the frequency of 1st flapwise vibration increases slightly because the increase of the rotor rotating speed causes the increasing of centrifugal force which will influence the effective stiffness of rotor blade. The frequency of 1st edgewise vibration as shown in Figure 2.27 (b) is a little higher than that of the 1st flapwise vibration and not affected so much by the increasing of rotor rpm because the flapwise direction of the blade has the least stiffness and the edgewise has higher stiffness than those of flapwise direction (Rao 2007). Potential spots of resonance which are marked with circles show that the wind turbine blade will experience a resonance near the 150rpm under 2Ω and 3Ω excitation in Figures 2.27 (a) and (b) respectively. These two spots suggest that the operation of the wind turbine near this rotating speed should be avoided wherever possible.

Figure 2.27 Resonance diagram of rotor blade under flow induced deformation (a) 1st flapwise vibration and (b) 1st edgewise vibration

2.3.7 Numerical simulation study of fluid-structure interaction of wind turbine

2.3.7.1 Introduction

Load predictions for wind turbines are commonly based on engineering design methods which use overly simplified physical models. The wind engineering community has been working on aero-elastic stability problems for quite some time (Bjorck 1996). However, most of these works are focused on flapwise vibrations only, in which a variety of engineering-type dynamic stall models have been developed. For example, Simms' group (Simms et al. 2001) introduced a quasi-steady linearized flap/lead-lag model to study edgewise vibrations in terms of blade and wind turbine parameters (Petersen et al. 1998). Nevertheless, all engineering design tools currently employed are not capable of taking into account of dynamic stall phenomena as well as obtaining the detailed information.

Two recent comparative studies of unsteady flow and fluid-structure interaction solvers for wind turbine applications were performed by Chaviaropoulos's group with researchers from many countries, who found major uncertainty and efficiency problems with these sophisticated simulation tools (Chaviaropoulos et al. 2003). The wind tunnel experimental results of National Renewable Energy Laboratory (NREL) (Simms et al. 2001) were compared with 19 simulation packages, two of which were unsteady Navier-Stokes solvers. Simms et al. (2001) were surprised by the wide variation between their experimental results and the various code predictions. One three-dimensional Navier-Stokes code did the best job of predicting aerodynamic forces under the simplest upwind zero-yaw conditions. However, the investigators indicated that success with this CFD code tends to be uncertain, varying significantly and very much depending on the type of airfoil used. Simms et al. (2001) reported that they had very good results in this effort with the S809 airfoil (Tangler and Somers 1996) used by NREL, but had poor results with some other airfoils in the past.

Another problem was the very high capacity of computational power is required for even the simplest cases. However, Simms et al. (2001) suggested that the success with this CFD code did show the potential benefit in pursuing CFD and that further work in this area is warranted.

2.3.7.2 Numerical method and uncertainties

The project of Chaviaropoulos et al. (2003) consists of two parts: 3D Navier-Stokes rotor simulations and aero-elastic stability investigations. In the first part, simulation tools with different physical and numerical models were assessed and their predictions were compared with detailed wind turbine aerodynamic data. Especially in stall and post stall operation regimes, large variations in the results were found, most likely due to differences in turbulence models which had been used. In the second part, based on Navier-Stokes codes coupled with structural solvers were employed to tackle the dynamic elastic problem, and their results were compared. For both the classical and the stall-induced flap-lag flutter problems, most of the results were in fair agreement. However, for these relatively simple problems, significant differences were found in response frequency and aerodynamic damping, probably caused by numerical methods used. Furthermore, for the stall-induced flutter problem, differences in lift and drag coefficients at high angles of attack are quite large, again due to uncertainties in the turbulence models used. It was found that this work demanded significantly large computer resources because each run required an order of few CPU days to a week on a fast workstation. Besides uncertainties from computer programming and round-off error, although these are not the main focuses of the present work, it is worth discussing them.

The uncertainties can be divided into two main groups. The most important group is the physical modeling errors which are those caused by inaccuracies in the mathematical model of the physics. These errors are different from the numerical method in nature. They involve

assumptions used in the physical models, initial and boundary conditions, as well as dimensionality assumption. It is generally believed that the physical model is responsible for large uncertainties in modern applied aerodynamics calculations. It is accountable for an important part of the scatter observed between experimental and computational data. Probably, the most crucial uncertainties for unsteady flow simulations of rotating machineries are the turbulence model being chosen. Finally, for the fluid-structure interaction, the uncertain structure properties and two-dimensional assumption may play a role. These uncertain model assumptions and boundary conditions lead to errors, in computed loads. In 2010, Min and Sankar (2010) proposed a hybrid method that combines the best features of vortex-capturing and vortex-fitting techniques for efficient vortex-rotor interaction simulation for experimental HART-II rotor model. It is based on Navier–Stokes/free-wake methods, which solve the flow around the rotor using Reynolds Averaged Navier–Stokes equations while capturing the far wake using discrete free-wake models. Since only the near field is modeled using the computationally expensive Navier–Stokes solver, the computational domain is much smaller than that for a full Navier–Stokes simulation. These methods give reasonable load predictions for steady level flight and hover. Compared with the traditional engineering method, this approach can include more flow physics.

Ferziger and Peric (1999) have included discretisation and solution errors estimation in their numerical modeling. Discretisation errors are those caused by the numerical replacement or discrete mapping of the physical model equations and boundary conditions into algebraic equations. Solution errors refer to the errors in the computer solution of the complete set of discrete, i.e. algebraic equations. Errors can be reduced using additional computational power and memory space. When the temporal and spatial step sizes are decreased, the discretisation error of a consistent scheme is reduced. Furthermore, when an iterative scheme is converging, additional iterations will reduce the convergence error (Versteeg and Malalasekera 2007).

With the current state of numerical methods and computing speeds, numerical errors can often be reduced to a satisfactory level for simple steady flow applications. For complex 3D unsteady flows and fluid-structure interactions, however, this is not the case. Due to the higher computational costs required for these simulations, temporal and spatial scales might be too expensive to resolve. In addition, the iterative convergence error has at all times to be smaller than the temporal discretisation error. Therefore, for the highly non-linear complex 3D unsteady flow and fluid-structure interaction simulations of wind turbines, numerical modelling uncertainties are non-negligible and should be addressed. 3D unsteady flow computations for high speed rotating machineries, which have Reynolds numbers in the order of millions and have many different spatial and temporal scales, implicit discretisation in time is most often used. As a result, the choice of the time step is not limited by stability restrictions coming from scales which do not require to resolve and can be solely driven by the accuracy desired to resolve the significant scales (Roache 1997). For implicitly discretised equations the efficiency of the flow solver strongly depends on the iterative solution technique. As the number of unknowns for the applications is large (in the order of millions), the iterative solution process is a very expensive part of the simulation. Efficiency gains in the iterative solver are therefore becoming significant.

2.3.8 Hybrid RANS/LES simulation method for wind turbine

2.3.8.1 Characteristic of turbulent flow

To solve the unsteady Navier-Stokes equations indicate that overall flow field have to be resolved from the large scale to the smallest Kolmogorov scale. It is equivalent to mean that the spatial discretization interval should be smaller than Kolmogorov scale. The dimension of the flow field domain should be big enough in order for the large vortex to be fully developed. A typical standard for an isotropic turbulent flow is that the domain size should be

at least fifty times larger to exclude the nonphysical correlations. Another boundary of the maximum time step is provided by the time scale of the smallest eddies. This limitation implies that the fluid will not be allowed to move more than one mesh spacing during one time step in order to obtain the relatively accurate results (Sagaut et al. 2013).

All the turbulent flow characteristics are closely related to the turbulent kinetic energy which is mainly generated with internal force which is caused by the largest scale eddies in the flow. The turbulent kinetic energy then will be transferred to the small scale eddies by the inviscid nonlinear interaction of eddies and then finally will be transformed by the viscous dissipative processes into heat. This procedure is illustrated in Figure 2.28 that the flow domain can be divided into three typical regions according to the way of energy transformation (Rodi 2013). The first region is made of the large scale of eddies and the turbulent energy is created due to the mean flow instability. Most of the large scale eddies are associated with the mean flow and rely on the turbulence generation mechanism. Therefore, these eddies do not hold a global characteristic. For those large scale eddies which are not related to the direction of turbulence generation, they obtain energy from the large energetic eddies via the mechanism which is called the inverse energy cascade. In the second region, these eddies will possess the intermediate size and transfer to small scale eddies by nonlinear interaction. The third region consists of the smallest scale eddies in which the viscous effect will become dominant and the kinetic energy will be dissipated into heat finally. The main reason for the energy transformation is the vortex stretching mechanism. Under this mechanism the vortex will go through a stretching process along their rotating axis and become longer in this direction. This kind of deformation will lead to the vortex diameter keeps decreasing while the volume remains the same. The vortex stretching mechanism will ensure that the vortex angular momentum is a constant and cause the vorticity in axial direction keep increasing

simultaneously. As a combined result, the kinetic energy will be transferred toward small eddies because the diameter of the vortex has been decreased (Sagaut et al. 2013).

Figure 2.28 Energy cascade of flow field

2.3.8.2 Comparison of RANS model and LES model

RANS turbulence model can be used to predict the flow field development for engineering purpose with enough accuracy. The flow should be relatively simple and the main flow need to be largely unidirectional and attached. The RANS turbulence model remains an important tool for cost effective engineer usage which is not only can be applied in research institution but also can be employed for field study in flow calculation (Dewan 2011). However, it is found that RANS models cannot deal with complicated flow where large scale anisotropic structures are dominant as well as the unsteady effects are prominent such as the situations of fluid-structure interaction. On the other hand, the LES turbulence model could handle these cases very well. However, the LES model is much expensive compare to RANS model because it always has to involve three dimensional and time variant. The near wall treatment

of the mesh also requires a huge amount of the grid nodes for a high resolution in the near wall circumstances. Therefore, the LES method is not applicable to the cases with high Reynolds numbers such as wind turbine rotor near stall condition.

The different characteristics of these two turbulence models show that they both have advantages and disadvantages. The weakness of each turbulence model method could be overcome by combining the two models together. The LES model could be only applied when the flow can not be resolved by RANS model due to the complexity or wherever the detailed information of the dynamic behavior of the flow field are required such as the high angle attack separation flow during wind turbine operation. While at the same time, the other zone of the flow field which is close to fully developed channel flow, the RANS turbulence model still can be used to carry out the calculation to obtain a reasonable accurate result. Therefore, such an approach will only use the LES method to calculate a part of the whole flow field where accuracy is the main concern and the other parts will be calculated with RANS method for economically use of the resources and computational time. This kind of method is called hybrid RANS/LES turbulence method. Figure 2.29 illustrates the classification of turbulence model.

Figure 2.29 Classification of computational fluid dynamic turbulence model

For Figure 2.29 it can be observed that from DNS to RANS, the simulation accuracy and computational time keep decreasing. From RANS to DNS the computational time and degree of freedom will increase. Therefore, a balance point of accuracy and cost should be selected near around hybrid RANS/LES model. Figure 2.29 demonstrates how the large scale eddies extract energy from main stream and break down into small eddies and finally become dissipation of energy. It also shows the different flow field when the large scale eddies are dominant then the LES model is applied and when the small eddies are prominent then the RANS turbulence method should be employed.

Based on the different ways of combining the RANS and LES turbulence models, there are several different kinds of Hybrid RANS/LES model such as embedded LES model, detached eddy simulation (DES) model and scale-adaptive simulation (SAS) model. Each hybrid model has its own strength and weakness. To find out the most suitable hybrid model, each of

simulation results with these hybrid models should be compared with experimental data to check its accuracy. Figure 2.30 demonstrates the different simulation results using different turbulence model.

Figure 2.30 (a) shows that the URANS model could not handle the large scale separation flow, since it is a Reynolds average method, the detailed information of the vortex has been ignored and a lot of flow field information has lost. Figure 2.30 (b) illustrates that the hybrid SAS model could precisely capture most of the information of the flow field from large scale eddies to relatively small eddies and provide a detailed information of the whole flow field at a reasonable cost. Therefore, in the present study, the hybrid RANS/LES turbulence model will be employed to investigate the aeroelastic stability of wind turbine.

**Figure 2.30 Vortex shedding from cylinder using different turbulence method (a) URANS
(b) SAS (Ansys 2015)**

2.4 Chapter summary

In this chapter, a review of wind energy and detailed wind turbine rotor design theory are introduced. A comprehensive literature review of basic theory of fluid-structure interaction and the aeroelastic stability of wind turbine are presented. The numerical simulation study of fluid-structure interaction of wind turbine including the numerical method and the advantages and disadvantages of using the hybrid RANS and LES turbulence models are introduced and discussed in details.

CHAPTER 3 WIND TURBINE ROTOR DESIGN AND FSI NUMERICAL SIMULATION CONFIGURATION

3.1 Wind turbine rotor design

The first step in a wind turbine blade design is to specify some basic parameters of the rotor, for example, the number of blade, the tip speed ratio, the rotor diameter, and etc. It is then following with the determination of the aerodynamic parameters of blade such as selection of airfoil, chord and twist angle. The designed wind turbine should have good performance under the rated working condition and operate satisfactorily and safely under the non-rated working scenario.

3.1.1 Wind turbine rotor parameters selection

1. Wind turbine rated power

The wind turbine blade length is proportional to the wind turbine rated power. Taking into account that the present work is a two way FSI simulation with hybrid RANS and LES turbulence modeling which will not only require lots of computational capacity but also imply that the computational time will increase exponentially due to large blade size. The rated power of wind turbine is then selected as 5kW, since the present low power capacity is good enough for conducting FSI numerical simulation using the current limited computer capacity of the software and hardware.

2. Tip speed ratio

The tip speed ratio has a close relationship with power coefficient. For a modern high speed wind turbine which is not operating under a stall status, the selection range of the tip speed ratio is usually ranging from 6 to 8 which could ensure the wind turbine to achieve high

performance (Miller 1997). As discussed earlier in Chapter 2, Section 2.1.2.1 the tip speed ratio in the present work has been chosen as 6.

3. Number of blade

As discussed earlier in Chapter 2, Section 2.1.2.1 the number of blade in the present work has been chosen as 3. For a modern HAWT the number of blade is normally chosen as 3 due to structure simplicity, relatively low system cost, symmetrical rotating force distribution, stable power output and low maintaining cost (Duquette and Visser 2003).

4. Rated wind speed

The rated wind speed will be selected as 10m/s based on Weekes and Tomlin (2013) to maximize the capture of the wind energy by the wind turbine and extended stable working range. Wind speed of 10m/s is also a common rated speed for a lot of wind turbines in operation and can ensure the simulation results reflect what happens in reality.

5. Power coefficient

For modern high speed HAWT, the power coefficient C_p is usually chosen around 0.45 (Zhao and Xu 2011) to conduct the estimation work of the blade length during the designing stage. The density of air ρ is assumed as 1.25kg/m³. The pressure is assumed as one standard atmosphere pressure which equals to 101,325Pa.

6. Rotor radius

The rotor radius is estimated using the following empirical equation (Burton et al. 2011):

$$R = (1/2) \sqrt{8P\eta/C_p\rho V_1^3\pi} \quad (3.1)$$

where P is the power produced by wind turbine, $\eta = 0.9$ is usually chosen as the mechanical and electrical component efficiency of wind turbine. Therefore, the radius of the rotor is determined as $2.66m$. The hub radius of the wind turbine is selected as $0.2m$ (Burton et al. 2011).

7. Angular velocity of wind turbine

Since the wind turbine rated wind speed is selected as $10m/s$, then to maintain the wind turbine to operate around the optimal tip speed ratio of 6, the angular velocity of wind turbine can be decided by the following equation:

$$\omega = \lambda V_1 / R \quad (3.2)$$

It is found that the determined wind turbine rotating angular velocity is $22.56rad/s$ or $215.43rpm$ (Burton et al. 2011).

8. Airfoil selection

The airfoil selection is very important to the wind turbine power efficiency. The deployment of suitable airfoil at different rotor radial location is the key step to ensure the designed wind turbine could deliver an expected power output. Normally from the tip ($r=R$) to the location $r/R = 0.30$ is the main power generation zone and the airfoil selected for this region should be thin with high lift to drag ratio to capture more energy. From the location $r/R = 0.30$ to the root, the airfoil selected for this region should be thick enough to provide the blade structure support. Beside the requirements mentioned above the aerodynamic characteristics change from the root to the tip should be continuous to ensure the relatively even load distribution on blade (Tangler and Somers 1996). The S-series airfoils are dedicated to design modern high speed HAWT by the academic institution of National Renewable Energy Laboratory (NREL). This airfoil family achieves low drag in clean condition and has a

maximized lift coefficient which is basically insensitive to surface roughness. Table 3.1 lists the S series airfoils which are suitable for different size of HAWT recommend by NREL (Tangler and Somers 1996).

Table 3.1 NREL airfoil family (Tangler and Somers 1996)

Blade Lengths (Meter)	Generator Size (kW)	Thickness Category	Airfoil Family (root-----tip)			
1-5	2-20	thick		S823		S822
5-10	20-150	thin		S804	S801	S803
5-10	20-150	thin	S808	S807	S805	S806
5-10	20-150	thick		S821	S819	S820
10-15	150-400	thick	S815	S814	S809	S810
10-15	150-400	thick	S815	S814	S812	S813
15-25	400-1000	thick		S818	S816	S817

According to the recommendation of Table 3.1, the airfoil S823 is selected to deploy from root to $r/R = 0.30$ and the airfoil S822 is selected to deploy from $r/R = 0.30$ to tip. Although according to Wood (2011) the thick airfoils not perform very well at the low Reynolds number, the S822 and S823 airfoil can maintain the stiffness and stability of the structure of the wind turbine blade under stall working condition. Therefore, the airfoils S822 and S823 are chosen to form the wind turbine blade. The details dimension of airfoil S823 and S822 can be found in Appendix A.

3.1.2 Mathematical modeling of blade aerodynamic characteristic

The Wilson and Lissaman design method (Wilson and Lissaman 1974) is a high precision method and widely adopted by wind turbine engineers because this method not only

introduces the axial and rotational factors but also includes the tip and hub loss factors. The Wilson and Lissaman design method consists of mathematical equations (2.29), (2.30), (2.31), (2.38), (2.39), (2.40), (2.41), (2.42) and (2.43), which were discussed earlier in Chapter 2. As the basic specification of wind turbine have been chosen, equation (2.31) shows that the relative wind direction is only related to axial induced factor and rotational induced factor.

From Figure 2.16 in Chapter 2 (Page 29) the following relationship between angles are shown below:

$$\phi = \theta + \alpha \quad (3.3)$$

Then solve the mathematical model in terms of a , b , and ϕ which will maximize the implicit function, dC_p , under several constrains which are shown as following for ease of reference.

$$\text{Max: } dC_p = (8F_T b(1-a)\lambda_r^3/\lambda^2)d\lambda_r \quad (2.43)$$

$$\begin{array}{l} \text{Subject to} \left\{ \begin{array}{ll} \tan \phi = (1-a)/(1+b)\lambda_r & (2.31) \\ b(1+b)\lambda_r^2 = a(1-aF_T) & (2.42) \\ Re = V_1(1-a)c/(v \sin \phi) & (2.41) \\ a/(1-a) = Bc(C_l \cos \phi + C_d \sin \phi)/(8\pi r(\sin \phi)^2) & (2.29) \\ F_T = F_{tip}F_{hub} & (2.38) \\ F_{tip} = (2 \cos^{-1} e^{-(B(R-r)/2R \sin \phi)})/\pi & (2.39) \\ F_{hub} = (2 \cos^{-1} e^{-(B(r-r_{hub})/2r_{hub} \sin \phi)})/\pi & (2.40) \end{array} \right. \end{array}$$

After obtaining the axial factor a , rotational factor b and relative wind direction angle ϕ , these parameters were input into equation (2.41), which lead to a rough guess of both the chord c and the corresponding Re . Then go to the airfoil polar drawing to get the corresponding lift and drag coefficients at this Re and put these two values into equation (2.29) to calculate a new chord. The difference between the new chord and the old one will be evaluated. If the difference is bigger than the convergence criteria then continue the iteration or otherwise accept the new value of the chord as the final result. The chord found will be input into equation (2.41) and then check the polar drawing of S822 and S823 to evaluate the angle of attack which will maximize the value of C_l . As mentioned in Chapter 2, Section 2.2.1, a buffer will be required to prevent or at least delaying the wind turbine slip into stall condition. Then the corresponding angle of twist will be obtained using equation (3.3).

According to the BEM method the blade has been divided into ten sections to search for the maximized value of power coefficient for each section. A program is written using the scientific computational software Mathematica (Zimmerman and Olness 2002) to solve the mathematic model mentioned above for each section. The computation results of blade twist angle and chord are shown in Figures 3.1 and 3.2 respectively. Figure 3.1 demonstrates that the blade twist angle decreases from root to tip which matches the prediction mentioned in Chapter 2. Figure 3.2 illustrates that the chord has a jump at the root and then decreasing smoothly from the root to tip. The jump is due to the lift coefficient transition between the two different types of airfoil used.

Figure 3.1 Twist angle distributions

Figure 3.2 Chord length distributions

According to the results obtained in Figures 3.1 and 3.2, airfoil coordinates of S822 and S823 with the wind turbine specification will be used to generate the three dimensional drawing using professional engineering drawing software SolidWorks2016 (Verma and Weber 2015). The blade element twist angle variation, blade isometric view and rotor isometric view are shown correspondingly in Figures 3.3, 3.4 and 3.5.

Figure 3.3 Blade element twist angle variation (from root to tip)

Figure 3.4 Isometric view for wind turbine blade

Figure 3.5 Isometric view for wind turbine rotor with hub

3.2 Configurations of numerical simulation for two-way FSI on wind turbine

3.2.1 Simulation tool introduction

Ansys is a finite element analysis software which can offer comprehensive engineering simulation solution. Ansys Fluent is one of the important solution components in the Ansys Workbench for solving the engineering related fluid problems and issues. Ansys Fluent is a high-performance CFD software tool which can provide trustable, precise, fast and stable solutions for a wide range of CFD and multi-physics applications. Fluent is well known for its excellent accuracy, stability and fast speed in computation especially for rotating machinery such as pumps, fans, compressors and wind turbines. The solution obtained from Fluent is highly reliable which has been verified by different industrial applications (Ansys 2015). Another powerful tool in structural analysis is Ansys Transient Structure Component which can deliver robust general analysis with stress, deformation, modal and fatigue simulations in fast and accurate way. It has been widely employed by a lot of mechanical

engineering companies in wind turbine industry to conduct the vibration and modal simulation before mass production. This structure analysis tool can provide a high reliable simulation result in a relatively short time and only request a small and affordable computer hardware capacity.

The latest Version 15 of Ansys integrates the new function of System Coupling which can provide a whole new multi-physics platform that can offer a fast and convenient approach to build up a robust link between the fluid analysis component obtained from Fluent and the structure analysis component from Transient Structure (Lawrence 2012). With this new multi-physics integrated platform the two-way FSI investigation of wind turbine can be quickly and conveniently conducted in a reliable way.

The new version Ansys 15 also provides an interface which can directly read the three dimensional model generated from SolidWorks. Therefore, the wind turbine three dimensional model generated by SolidWorks could be very conveniently imported into Ansys 15 to carry out the fluid-structure interaction numerical simulation in both Fluent and Transient Structure components.

3.2.2 Flow domain configuration

3.2.2.1 Turbulence model selection

As suggested by Menter and Egorov (2010), adapting the SAS (Scale Adaptive Simulation) concept to the $k\omega$ -SST (Shear Stress Transport) model (Menter 1994) can overcome the major disadvantage of two-equation turbulence model such as the $k-\epsilon$ model. The main reason is due to that the derivation of the turbulence scale equation of $k-\epsilon$ model including specific dissipation rates ω and ϵ , are mainly depending on the dimensional scale (Versteeg and Malalasekera 2007). The $k\omega$ -SST turbulence model could provide a more rigorous beginning location in the transportation equation with the integral length scale of turbulence.

Therefore, the $k\omega$ -SST hybrids with SAS will be employed in the present work to implement the two-way FSI numerical simulation.

$$\partial(\rho k)/\partial t + \text{div}(\rho k v) = \text{div}[(\mu + \mu_t/\sigma_k) \text{grad}(k)] + P_k - \beta^* \rho k \omega_T \quad (3.4)$$

$$\text{where } P_k = 2\mu_t S_{ij} \cdot S_{ij} - (2\rho k \delta_{ij}/3)(\partial U_i/\partial x_j)$$

$$\begin{aligned} \partial(\rho \omega_T)/\partial t + \text{div}(\rho \omega_T v) = & \text{div}[(\mu + \mu_t/\sigma_{\omega_T,1}) \text{grad}(\omega_T)] + P_{\omega_T} - \beta_2 \rho \omega_T^2 + \\ & (2\rho/\sigma_{\omega_T,2} \omega_T)(\partial k/\partial x_j)(\partial \omega_T/\partial x_j) \end{aligned} \quad (3.5)$$

$$\text{where } P_{\omega_T} = \gamma_2(2\rho S_{ij} \cdot S_{ij} - (2\rho \omega_T \delta_{ij}/3)(\partial U_i/\partial x_j))$$

The physical meaning of each term in equations (3.4) and (3.5) is as follow:

Rate of change of k or ω_T	+	Transport of k or ω_T by convection	=	Transport of k or ω_T by turbulent diffusion	+	Rate of production of k or ω_T	-	Rate of dissipation of k or ω_T
--	---	--	---	---	---	---	---	--

Where v is velocity vector, μ is dynamic eddy viscosity, k is turbulent kinetic energy per unit mass, $\omega_T = \varepsilon/k$, S_{ij} are limiters to give improved performance in flows with adverse pressure gradient and wake region, the turbulent kinetic energy production is employed to prohibit the build-up turbulence in stagnation regions. U_i is velocity tensor, x_j is gradient tensor, δ_{ij} is Kronecker delta given by $\delta_{ij} = 1$ if $i = j$ and $\delta_{ij} = 0$ if $i \neq j$, σ_k , $\sigma_{\omega_T,1}$, $\sigma_{\omega_T,2}$, γ_2 , β_2 and β^* are constants with values of 1.0, 2.0, 1.17, 0.44, 0.083 and 0.09 respectively (Versteeg and Malalasekera 2007)

The $k\omega$ -SST model is a hybrid model which divides the flow field into two regions. Around the near wall zone the standard $k\omega$ model will be utilized and the standard $k\varepsilon$ model is employed in the fully developed turbulent zone far from the wall. The $k\omega$ -SST model can deliver a good result for the adverse pressure gradient boundary, zero pressure gradient

boundary and free shear layer flow which can provide high accuracy for non equilibrium turbulent boundary layer flow of wind turbine (Blazek 2001).

3.2.2.2 Turbulence model verification

To ensure the hybrid RANS/LES turbulence model could be employed in the present wind turbine FSI research, the experimental data from NREL Phase VI two-blade wind rotor will be used to validate the suitability of this hybrid turbulence model. In this experiment, a two-blade HAWT which has twisted and tapered blade in 10.06m length with a rated power of 19.8kW was carried out in the NASA Ames wind tunnel (Hand et al. 2001). This is one of the most integrated, precise and reliable full size wind turbine experiment in a wind tunnel. This experiment data is studied by many wind turbine and computational researchers and has been used to validate the wind turbine simulation results and improve the accuracy of aerodynamic prediction (Pape and Lecanu 2004). The NREL phase VI experiment uses the S809 airfoil to form the wind turbine blade. Some selected blade cross-section data are listed in Table 3.2.

Table 3.2 Part of cross section data from NREL Phase VI experiment

Radial Distance (m)	Chord (m)	Twist angle (degree)	Airfoil type
0.51	0.22	0.00	Cylinder
1.51	0.71	14.29	S809
2.34	0.63	4.72	S809
3.19	0.54	1.12	S809
4.02	0.46	-0.38	S809
4.78	0.38	-1.47	S809
5.03	0.36	-1.82	S809

Figure 3.6 (a) Wind turbine installed in wind tunnel (b) photo of wake flow visualization (Hand et al. 2001)

Table 3.2 shows that the NREL Phase VI wind turbine blade chord decreases along the blade radial direction after $r=1.51\text{m}$. The twist angle keeps increasing from radial distance 0.51m to 1.51m , after that the twist angle keeps decreasing as blade radial distance increasing. From root to location of 0.51m the blade uses the shape of cylinder and from 0.51m to tip employs S809 airfoil. Figures 3.6 (a) and (b) show a full scale two-blade HAWT mounted in a wind tunnel and the flow visualization of the wake using the digital camera respectively.

To validate whether the hybrid RANS/LES turbulence model is suitable for predicting the aerodynamic performance of wind turbine, several simulations have been conducted at different locations of the NREL Phase VI wind turbine blade under different wind speeds. The present simulation results of pressure coefficient are being compared with the experiment data and are shown in Figure 3.7 for two dimensionless radial locations, $r/R=0.8$ and $r/R=0.46$ under two different wind speeds of 7m/s and 25m/s (Hand et al. 2001).

(a)

(b)

(c)

(d)

Figure 3.7 Comparison of the present simulation results with experimental data of C_p (a) 7m/s and (b) 25m/s at location of $r/R=0.8$, (c) 7m/s and (d) 25m/s at location of $r/R=0.46$

Figure 3.7 (a) illustrates that at wind turbine location of $r/R=0.8$ under the wind speed of 7m/s, the present simulation results of pressure coefficient have matched with those of the experimental data very well except near the location of upper surface around $0.35 \leq x/c \leq 0.42$. Figure 3.7 (b) shows that at location $r/R=0.8$ under the wind speed 25m/s at which the wind turbine is in stall condition, the present simulation results of pressure coefficient agree with those of the experimental results except the lower surface around $0.01 \leq x/c \leq 0.43$. Figure 3.7 (c) illustrates that at location of $r/R=0.46$ when the wind speed is at 7m/s, the present simulation results of pressure coefficient agree with those of the experimental data very well except the part near around the upper surface at $x/c = 0.02$. Figure 3.7 (d) shows that at location of $r/R=0.46$ under the wind speed 25m/s the present simulation results of pressure coefficient agree with the experimental pressure coefficient results except the location at lower surface near around $x/c = 0.01$. Figures 3.7 (a), (b), (c) and (d) show that the present hybrid method of SAS with $k\omega$ -SST can deliver reasonable accurate simulation results under different wind speed situation for wind turbine aerodynamic performance prediction of pressure coefficient and therefore can be applied in the present study for further investigation (Hand et al. 2001).

3.2.2.3 Computational domain setup

The shape of the whole computational flow domain is cylindrical in shape which includes the total wind turbine rotor as shown in Figure 3.8. The total length of the stationary domain is 8 times of rotor radius from inlet to outlet in axial direction to ensure that the wake behind wind turbine rotor is fully developed. In radial direction the flow domain will extend from the centerline of the hub to the boundary of the cylinder which is 3 times of rotor radius to keep a low blockage ratio so as to reduce the wall effect according to Maskell (1963). The wind turbine rotor is placed at the location of a rotor diameter from inlet and the center line of the rotor is concentric with the flow domain. Bak et al. (1999) showed that the effect of tower

could be neglected for a upwind wind turbine under a relative low wind speed situation. To precisely capture the flow field variation near the wind turbine blade and satisfy the SST-SAS turbulence modeling requirement, a rotating domain with fine mesh will be employed as shown in Figure 3.8. The stationary domain and the rotating domain will be connected with mesh interface and the sliding mesh method between these two domains will be used in the simulation work. To simplify the model employed and save computational time, the wind turbine tower is not included in this project.

Figure 3.8 Flow field domain and boundary conditions

The inlet is set at nominal wind speed with 1% of turbulence intensity (Ansys 2015). The static pressure of 1 atmosphere is applied at the outlet. The no slip and smooth surface conditions are applied on wall boundary. Based on the recommendation of Ansys (2015) the left surface of cylinder beside the inlet and outlet is set as flow out boundary to ensure the calculation process converging quickly. The blade will rotate around Z axis in anti-clockwise direction with fixed angular velocity of 22.56rad/s. This rotating speed is equivalent to a

nominal wind speed of 10m/s at designed tip speed ratio 6. In this two-way study, four different wind speeds will be tested and simultaneously at each tested wind speed the rotating speed is kept constant in order to observe the wind turbine power performance and blade vibration under different tip speed ratio. The convergence criteria for the fluid domain is set as 10^{-3} . With the consideration that the turbulence model employed is SAS-SST and the blade is rotating at angular velocity of 22.56rad/s, then the time step will be set as 5×10^{-3} s (Benra 2006) and the total simulation time will be set as 3s to achieve a balance between the reasonable accurate result and the computational time used.

3.2.2.4 Computational mesh setup

Unstructured mesh will be employed in flow field domain in the present study. As shown in Figures 3.9 and 3.10 about 6.4 million elements including tetrahedral, prism and pyramid have been created for the rotor and the whole flow field using Ansys Fluent meshing. The ratio of the element to node is around 5 with a tetrahedral dominant unstructured mesh. The unstructured mesh near the blade and hub surface will expand with a growth ratio of 1.1 is shown in Figure 3.9. The minimum length of the element in global is 0.01m. This minimum length is necessary because the airfoil shape consists of high curvature surfaces which cause difficulties in capturing all the details of the blade aerodynamic characteristic.

Figure 3.9 Mesh growth rate around the rotor surface is 1.1 in flow domain

To precisely obtain the detailed information of the viscous vortex of the flow field, an inflation mesh layer becomes necessary. Boundary layer effects lead to velocity gradient which is normal to the blade surface, therefore, the suitable elements fulfilled this requirement have to possess a high aspect ratio (Ansys 2015). Figure 3.10 demonstrates the inflation layer employed in flow field for this study. Near the boundary, sixteen layers of inflated meshes are used to generate the prism and pyramid elements with the growth rate of 1.2 to capture the viscous effect of the flow field. Due to the capacity limit of the work station employed, the mesh should not be finer than those used in inflation mesh, otherwise the memory of computer will not be sufficient enough to sustain the compacity. Therefore, the inflation mesh configuration is relatively fine in terms of current computer hardware, but acceptable for the simulation. This is one of the disadvantages of the whole flow field simulation which will consume a lot of computer resources.

Figure 3.10 Inflation layers around the wind turbine blade in flow domain

The requirement of precise stimulation of the flow field demands at least a minimum of 10 grid nodes inside the boundary layer. In Ansys Fluent, there is an automatic wall function which is used to deal with the boundary conditions. This function will automatically switch from the low Reynolds number approach to a wall treatment, according to the grid space

which is input by the user. One of the major merit of the wall function is that the high gradient shear layer around the wall will be treated by relatively coarse mesh to save the memory and CPU time of the computer (Lawrence 2012).

In terms of the mesh quality, the skewness of the element is defined as difference between the cell shape and the shape of an equilateral element of equivalent volume. The smaller the skewness, the better the quality of the element. For the purpose of obtaining an accurate result, the ideal situation is to have only small number of elements with skewness bigger than 0.5 (Lawrence 2012). For the present work, the elements with skewness larger than 0.5 are less than 1.6 percent in the whole flow field domain. Therefore, the mesh quality is good and acceptable.

3.2.2.5 Mesh independence investigation

To ensure the CFD simulation results will not be different due to the different mesh size and amount, a mesh independence investigation have been conducted. Four kinds of different mesh size have been employed to obtain the wind turbine power coefficient under 10m/s. The configurations of the flow domain of these four different mesh size simulations are the same as described in previous paragraph. The results are shown in Table 3.3.

The simulation data in the table show wind turbine power coefficient under four different mesh amounts. For the mesh amount of 4.81 million and 5.74 million the difference from the standard power coefficient value is more than 5% and is not accurate enough. For the mesh number of 6.43 million and 7.16 million the difference is less than 2% and is qualified for the accurate simulation works. Since power coefficient variation is not much between 6.43 million and 7.16 million meshes, then with the consideration of saving computational time the mesh amount of 6.43 million is adopted in the present work.

Table 3.3 Verification of mesh independence

Mesh Amount (Million Elements)	4.81	5.74	6.43	7.16	Designed Power Coefficient 0.470
Power Coefficient	0.438	0.444	0.461	0.463	
Difference	0.032	0.026	0.009	0.007	
Percentage	6.81%	5.53%	1.91%	1.49%	

3.2.3 Mechanical structure configuration

Due to high curvature shape of the wind turbine rotor, the unstructured mesh is utilized to capture the sharp edge and large amount of curved surfaces. The mesh elements are tetrahedral which are generated in Ansys Mechanical Component. The minimum size and the growth rate of the element are 3mm and 1.85 respectively. The total amount of the meshes for the elements is around 0.7 million. In order to obtain the deformation and vibration frequency of the structure accurately, the finer mesh is used around the trailing edge and the tip of the blade. Note that blade at these locations are thin in order to sustain high aerodynamic and gravity forces. The root of the blade and the hub of the rotor consist of a relative solid structure and the coarse meshes will be used. The meshes for the structure analysis of blade and rotor are shown in Figures 3.11 and 3.12 respectively.

Figure 3.11 Blade mesh around tip area in structure domain

Figure 3.12 Rotor mesh around root area in structure domain

For Mechanical Analysis Component, the results of equivalent stress, total deformation and up to sixth order of vibration mode will be obtained. The fiber glass which is commonly used for small size wind turbine blade will be employed as the rotor material in this numerical simulation work (Brondsted and Nijssen 2013). The property of material of wind turbine blade including the data of Young's modulus is very important to blade structure analysis (Sessarego et al. 2014). Since the wind turbine blade is designed as solid and the fiber glass is

chosen as blade material then the data of Young's modulus of blade material will be selected as 72GPa. It is known that one of the critical structural loads on a small wind turbine blade is the gyroscopic moment caused by the blade yaw and this moment will be large particularly when the yaw rate is very high (Wilson et al. 2008). Although the effect of blade yaw is very important, the influence of yaw rate will not be included in order to simplify the configuration two-way FSI simulation and reduce the computational time.

3.2.4 Fluid-structure interaction configuration

The flow process of interaction configuration is shown in Figure 3.13

Figure 3.13 Process flow chart of two-way fluid-structure interaction

The whole fluid-structure interaction flow field analysis consists of three parts. Firstly, the whole flow domain will be analyzed in Fluent. Secondly, the pressure on the blade and hub surfaces will be transferred to Transient Structure Analysis Component to obtain the stress and total deformation. After that, the total deformation on the rotor will be transferred back to flow analysis component in order to perform the two-way FSI simulation. The two analysis components will be connected by the System Coupling Component which will in charge of the data transfer between the Fluent and the Transient Structure as well as storing the simulation results. The wind turbine geometry will be shared by the Fluent and Transient

Structure Component, and the System Coupling Component will be responsible for the coordination of the set up of the flow and structure domains (Ansys 2015).

There are two methods of fluid-structure interaction in Ansys. The first method is one way which means that there are only forces transfer from fluid to structure and no structural displacement transfers back to fluid. The other method is two-way which means that the force will be transferred from fluid to structure and the displacement of structure will be transferred back to flow domain. For a more in depth study, the present simulation work will be conducted as an unsteady two-way FSI. The simulation process will be conducted by parallel mode using the fast work station with four cores CPU. Even for this small size two-way case, it requires four days of CPU time by the advanced computer for one converging result. For the LES two-way case the CPU required will be hundred times of the hybrid case. The more accurate LES simulation process of unsteady two-way fluid-structure interaction will be processed in future by the Hyper Computer in NTU.

3.3 Chapter summary

In this chapter, the wind turbine rotor design process including the wind turbine rotor parameters selection and the mathematical modeling of blade aerodynamic characteristics are presented. The flow domain configuration, the structure domain configuration and the fluid-structure interaction configuration are all explained and discussed especially for the numerical simulation for two-way FSI on wind turbine.

CHAPTER 4 WIND TURBINE ROTOR FSI NUMERICAL SIMULATION RESULTS ANALYSIS

4.1 Numerical simulation results for two-way FSI on wind turbine

The results will be presented in two main parts. The first part is related to the flow domain which includes the velocity and pressure distributions, surface streamlines and velocity vectors. Several different wind speeds will be tested and the corresponding curve of pressure coefficient, pressure contour, streamline and power coefficient curves will be illustrated. The second part is mainly about the structural response under the different wind speeds. The contours of the stress, deformation, natural frequency and vibration mode of the turbine blade will be presented.

4.2 Flow domain simulation results analysis

4.2.1 Velocity streamline

Figure 4.1 demonstrates the streamline generated from the blade (including hub portion) and leaving the rotor surface to wake region for wind speed 13m/s. The vortex sheds from the blade surface and continues to develop thereafter. It can be observed that the shed vortex keeps increasing in both diameter and pitch along the streamline direction.

Figure 4.1 Velocity streamline for wind speed 13m/s

The streamline distributions on the blade lower surface under four different wind speeds of 10m/s, 13m/s, 16m/s and 20m/s are shown in Figures 4.2 (a), (b), (c) and (d) respectively. Note that the wind flows perpendicularly out from the paper. From the comparison of the streamline distributions under four wind speeds, it is clear that in Figure 4.2 (a) most of the flow remain attached to nearly the whole lower blade surface at 10 m/s wind speed. However, when the wind speed increases to 13m/s as shown in Figure 4.2 (b), the flow separation begins to emerge. It can be observed in the figure, that the flow has started to separate from $r/R = 0.06$ to $r/R = 0.96$, and the range of flow separation keeps increasing until dominates all the region of the blade surface when the wind speed has increased to 20 m/s as shown in Figure 4.2 (d). At the same time due to the presence of the vortex bubble the radial flow begin to emerge. These phenomenon are due to the increase of wind speed

which change the angle of attack and lead to the flow separation. The lift will drop dramatically because the blade has stepped into stall condition.

(a) Wind speed 10m/s

(b) Wind speed 13m/s

(c) Wind speed 16m/s

(d) Wind speed 20m/s

Figure 4.2 Streamline distributions on lower surface at wind speeds of (a) 10m/s, (b) 13m/s, (c) 16m/s and (d) 20m/s. Note that wind velocity V , flows out from the paper.

Figure 4.3 shows the streamline distribution on blade upper surface at four different wind speeds of 10m/s, 13m/s, 16m/s and 20m/s and Figure 4.4 shows an enlarge view of stagnation streamline on blade lower surface at the wind speed of 10m/s. From Figure 4.3 it can be observed that the position of the stagnation streamline is almost the same for all four cases. These results show that within 10m/s to 20m/s the wind speed variation has little influence on the location of stagnation streamline.

Figure 4.3 Streamline distributions on upper surface of wind turbine blade: (a) 10m/s, (b) 13m/s, (c) 16m/s, (d) 20m/s, flows out from the paper.

A typical example is shown in Figure 4.4 for an enlarge view for wind speed of 10m/s. It can be observed that the stagnation streamline is near the leading edge due to the angle of attack near the root region of the blade ($0 < r/R < 0.3$) has increased steeply and far beyond the optimal value from the BEM method (Gasch and Tvele 2012).

Figure 4.4 Enlarge view of stagnation streamline under wind speed of 10m/s.

4.2.2 Velocity vector

The velocity vectors which are used to describe the flow separation on the lower blade surface are shown in Figures 4.5 (a), (b), (c) and (d) for wind speed of 10m/s, 13m/s, 16m/s and 20m/s respectively. In Figures 4.5 (a), the wind turbine is rotating under the rated wind speed 10m/s, the velocity vectors demonstrate that there is no separation until the flow reaches nearly the end of the trailing edge. The attached flow on the lower blade surface ensures the maximized lift force is applied on wind turbine blade. In Figure 4.5 (b), the radial velocity begins to emerge as the blade is rotating under the stall condition when the wind speed increases to 13m/s. The flow begins to separate due to the vortex shedding effect, and the rotating centrifugal force induced by the rotating blade has switched the flow from tangential direction to the radial direction. It can be observed that the flow separates at about 2/3 of the blade width from the leading edge. In Figure 4.5 (c), as the wind speed increases, the blade works under a deep stall condition. The reverse velocity vector begins to emerge at

2/3 of the blade length because of the centrifugal effect from rotating blade is beginning to dominate in the region. Also the flow separation occurs much earlier at about 1/4 of the blade width from the leading edge. In Figure 4.5 (d), the radial and reverse flow velocity become dominant in the region of 1/3 of the blade length on the blade surface because the wind turbine is operating under a full stall condition. Under such condition, the lift force will decrease quickly and the drag force will increase dramatically due to the strong flow separation on the blade surface.

(a) Wind speed 10m/s

(b) Wind speed 13m/s

(c) Wind speed 16m/s

(d) Wind speed 20m/s

Figure 4.5 Velocity vectors on lower surface with wind speeds of (a) 10m/s, (b) 13m/s, (c) 16m/s and (d) 20m/s. Note that the turbine blade is rotating in clockwise direction.

4.2.3 Pressure contour

Figure 4.6 shows the pressure contour at four different radial locations of a wind turbine blade under wind speed of 10m/s. From Figure 4.6 it can be observed that from the root part of the blade to the tip part of the blade the pressure difference keep increasing which will result in corresponding lift force enforcement. The pressure difference of upper surface and lower surface at location $r/R=0.25$ is roughly equal to 1000Pa (see Figure 4.6 (a)), while the pressure difference is roughly equal to 5500 Pa at the location $r/R=0.95$ (please refer to Figure 4.6 (d)). The huge difference between these two values is due to the different angle of attack at different radial location. At $r/R=0.95$ as shown in Figure 4.6 (d) the flow is almost attached and there is no separation. At $r/R=0.75$ (see Figure 4.6 (c)) the flow separation has begun to appear while at $r/R=0.50$ (see Figure 4.6 (b)) the low pressure zone on upper surface has enlarged and can observe a flow separation area behind the airfoil section. At $r/R=0.25$

which is made of airfoil S822, the positive pressure is much bigger than the other location indicated the high drag force.

Figure 4.6 Pressure contour at different locations (a) $r/R=0.25$, (b) $r/R=0.50$, (c) $r/R=0.75$ and (d) $r/R=0.95$ of a wind turbine blade under wind speed of 10m/s

Figure 4.7 shows the pressure contour at four different radial locations of a wind turbine blade under wind speed of 16m/s. Figure 4.7 (d) shows that at $r/R=0.95$ the flow separation begins to appear behind the leading edge of airfoil section and the minimum negative pressure presents near around the stagnation point indicated with the dark blue colour. Figure 4.7 (c) demonstrates that at $r/R=0.75$ the detached flow zone moves towards the leading edge due to the bigger angle of attack. Figure 4.7 (b) demonstrates that at the location of $r/R=0.50$ the detached flow zone will start almost from the leading edge and spreads all over the upper surface. Figure 4.7 (a) demonstrates that at the location of $r/R=0.25$ the flow will slightly separated from the upper surface and the negative pressure zone are much more evenly distributed compared to the other three radial locations. This phenomenon is caused by the very high angle of attack near the root part and the characteristics of the thick airfoil S822.

Figure 4.7 Pressure contour at different locations (a) $r/R=0.25$, (b) $r/R=0.50$, (c) $r/R=0.75$ and (d) $r/R=0.95$ of a wind turbine blade under wind speed of 16m/s

Figure 4.8 shows the pressure contour at four different radial locations of a wind turbine blade under wind speed of 20m/s. Figures 4.8 (a), (b), (c) and (d) illustrate the similar case as the Figures 4.7 (a), (b), (c) and (d) respectively with only difference that flow separation and the drag force are even much stronger than those in Figure 4.7 due to the high wind speed of 20m/s.

Figure 4.8 Pressure contour at different locations (a) $r/R=0.25$, (b) $r/R=0.50$, (c) $r/R=0.75$ and (d) $r/R=0.95$ of a wind turbine blade under wind speed of 20m/s

With the comparison of pressure contours at different radial locations shown in Figures 4.6, 4.7 and 4.8, it can be observed and concluded that as wind speed increasing the flow separation will move more rapidly towards the leading edge of the airfoil section due to the high angle of attack. Although as wind speed increased the flow separation will move towards leading edge, the separation at $r/R=0.25$ is not prominent for all three wind speeds. This phenomenon is caused by the stall delay effect which is due to the high speed rotating wind turbine blade (Ronsten 1992). The stall delay phenomenon is especially remarkable for the root part because the rotating effect has much less influence on the attached flow zone. When the blade is rotating the detached point will move towards the trailing edge and result in the lift force increase and drag force decrease. The rotating blade will not only lead to the stall delay effect, but can also cause the elimination of the flow separation inside the

boundary layer near the root of the blade (Du and Selig 2000). The rotating effect will result in a high lift coefficient of a rotating airfoil than a stationary airfoil. The physical explanation is that within the rotating blade boundary layer surface the centrifugal force will cause the axial movement of the flow in the boundary which lead to a thinner boundary layer. At the same time the Coriolis force will lead to the decline in both the acceralation and the adverse pressure gradient in the chord wise direction. The combination of the centrifugal force and the Coriolis force will result in the flow seperation point moving towards the trailing edge. Currently the wind turbine design is based on the classic two dimensional BEM method and the two dimensional airfoil experiment data without the consideration of rotating effect. Therefore, the expected stall calculated with two dimensional theory will not be shown in the present three dimensional rotating study. Since the CFD turbulence model used in the present work is the whole flow field with SAS-SST which can simulate the real situation in the wind turbine industry, the present work should deliver a more relaiable and accurate results than the two dimensional and stationary works.

Figures 4.9 show the pressure contour at location $r/R=0.95$ of a wind turbine blade under 20 m/s wind speed at the computational time interval of 0.02s, 0.05s, 0.08s and 0.1s respectively. It can be observed from Figures 4.9 (a) to (d) that how the flow separation develops near around the leading edge within the investigating time. In Figure 4.9 (a) the flow separation begins to emerge at leading edge after the simulation starts in a short time of 0.02s. In Figures 4.9 (b) and (c) the flow separation zone develop to a big area within a time interval of 0.03s. In Figure 4.9 (d) the flow separation is fully developed near the leading edge. These figures demonstrate that the flow developing process occurs within a short time 0.1s but the simulation results provide the detailed information of flow field around blade section. These results have proved that the time step of 5×10^{-3} s is fine enough to capture the details of the flow field variation during the wind turbine FSI simulation.

Figure 4.9 Pressure contour at locations $r/R=0.95$ of wind turbine blade under wind speed of 20m/s at time (a)0.02s, (b)0.05s, (c)0.08s and (d)0.1s

4.2.4 Pressure coefficient

Figures 4.10 (a) to (f) show the pressure coefficient distributions at six different radial locations of wind turbine blade under rated wind speed of 10m/s. It can be observed that the incoming flow beyond the $r/R=0.70$ still attach to the wind turbine blade surface in the form of attached boundary layer as shown in Figure 4.10 (d) to (f) at this rated wind speed. The friction between the fluid and the blade surface will result in slowing down the flow and lead to different pressure at upper surface and lower surface. The pressure coefficient line will separate from the leading edge and merge at the trailing edge. Figures 4.10 (a) to (c) demonstrate that the maximum pressure difference near around the nose area of the airfoil section are much higher than those values of the high radial locations as shown from Figures 4.10 (d) to (f). This phenomenon is caused by the severe flow separation around the nose area due to the high angle of attack at these blade locations. The maximum pressure difference near around the leading point is not increasing as the dimensionless radial location, r/R , is reducing. The pressure difference at $r/R=0.4$ is less than that at $r/R=0.5$ because the airfoil used at the radial location in Figure 4.10 (a) is in transition from S823 to S822.

The different aerodynamic characteristics of airfoil S822 and S823 such as the chamber thickness, the chord and the aerodynamic center will lead to the different simulation results with the present wind turbine design and operating condition. Other than the different airfoil type, another reason of this decline of maximum pressure difference from suction surface and pressure surface is that the location on the blade in Figure 4.10 (a) is nearer to the root than that of Figure 4.10 (b). As discussed in previous paragraph, for the rotating blade the closer to the root, the flow will be more seriously affected by the stall delay phenomenon. Since the wind speed for Figure 4.10 is set as 10m/s which will only ensure the tip work under the designed angle of attack corresponding to the maximum lift coefficient, the location as shown

in Figure 4.10 (a) is within flow separation condition already which will be seriously affected by the stall delay phenomenon.

Figure 4.10 Pressure coefficients at six locations (a) $r/R=0.40$, (b) $r/R=0.50$, (c) $r/R=0.60$, (d) $r/R=0.70$ (e) $r/R=0.80$ (f) $r/R=0.90$ of wind turbine blade under wind speed 10m/s

Figures 4.11 (a) to (f) show the pressure coefficient distributions at six different radial locations of wind turbine blade under wind speed of 16m/s. From the comparison of the corresponding figures between Figures 4.11 and 4.10, the same trend of the maximum pressure difference between the upper and lower surfaces near around the leading edge can be observed. The pressure variation of the airfoil along the upper surface is much more severe than those along the lower surface. This is because the boundary layer along the lower surface is relatively thin and there is almost no flow separation which makes the boundary layer has little influences on the flow field as well as the pressure distribution. Another distinguished difference is the pressure difference between the leading edge of the upper surface and the lower surface is much higher than the other portion of the airfoil surfaces. This phenomenon is due to the large geometry curvature around the leading edge. For the incoming air passes around the highly curved leading edge of the airfoil will cause the wind speed to accelerate. According to the Bernoulli equation high flow velocity will lead to low pressure. The value of the pressure on upper surface in Figure 4.11(a) is higher than that in Figure 4.10 (a). It is because that airfoil under flow with large angle of attack, the boundary layer thickness will increase dramatically and causing the flow to separate from the upper surface. Another reason is that the increasing wind speed will correspondingly increase the local angle of attack of the airfoil. The increase of the angle of attack will lead to the increase of curvature of the streamline of the fluid flow around the airfoil leading edge and will result in generating much higher peak suction pressure peak value.

Figure 4.11 Pressure coefficients at six locations (a) $r/R=0.40$, (b) $r/R=0.50$, (c) $r/R=0.60$, (d) $r/R=0.70$ (e) $r/R=0.80$ (f) $r/R=0.90$ of wind turbine blade under wind speed 16m/s

Figure 4.12 shows how the pressure coefficient change at different time interval at a specific radial location $r/R=0.80$ under the wind speed 16m/s. The shapes of pressure coefficient, shown in Figures 4.12 (a), (b) and (c), keep evolving with the increasing time step from $t=0.02s$ to $t=0.05s$. This shape variation is due to the combination of the unstable fluid-structure interaction between the blade and the flow at the initial calculation stage of computation. From the time step of 0.07s as shown in Figure 4.12 (d) the shape of pressure coefficient curve is quite similar with slight variation from Figure 4.12 (c) at 0.05s. This is mainly due to the CFD calculation has reached the stable condition which also indirectly verify that the time interval and total time selected is fine and long enough in the present simulation. Although the shapes in Figures 4.12 (d), (e) and (f) are almost the same, there are still some small differences for example, the fluctuation of the maximum pressure value on upper surface close to the leading edge. These differences are caused by the unsteady flow around the wind turbine blade which is due to the vortex shedding and the fluid-structure interaction.

Figure 4.12 Pressure coefficients at location $r/R=0.80$ of wind turbine blade under wind speed 16m/s at different time (a) 0.02s, (b) 0.04s, (c) 0.05s, (d)0.07s, (e)0.09s and (f) 0.10s

4.2.5 Power coefficient

From the flow domain computational results, the torque applied on blade can be obtained and the power coefficient under different wind speed can be determined with equation (2.14). Figure 4.13 shows the power coefficient versus wind speed. Since the rated wind speed for the present 5kW wind turbine is 10m/s, the increase of wind speed will result in high angle of attack and lead the wind turbine blade operate under the stall condition. The maximum value of power coefficient is only 0.46, achieved at the wind speed of 10m/s, which is much lower than the Betz limit. The difference is due to drag loss and tip loss, because the blade design is not perfect. Although the Prandtl total loss factor has been taken into account in the blade design theory, the total loss factor is derived based on some simplified assumptions such as the periodic effect of the flow due to the limited blade number is ignored, the blade element in radial direction is independent from each other and so on. This type of total loss factor which is employed in BEM method will result in a wind turbine blade design will not match perfectly with the theoretical requirement. Beside the drag loss and tip loss, the stall also reduces the power coefficient due to the flow separation on the lower surface of the blade. The flow separation will generate a lot of vortices which will consume a large amount of energy from the wind and lead to a severe drop of wind turbine power coefficient. This could explain why the maximum value of power coefficient is much less than Betz limit. As the wind speed increase from 10m/s to 20m/s, it is clear from the figure that the power coefficient has dropped significantly. The reason is that when the tip speed ratio is less than 6, the wind turbine has already worked under stall condition with the stall stage progressing deteriorated, the flow separation will become dominant all over the wind turbine blade lower surface. The flow separation which increases the drag force and reduces the lift force will cause the wind turbine power coefficient to drop correspondingly.

Figure 4.13 Wind turbine power coefficient versus different wind speeds

Figure 4.14 demonstrates the comparison of the 5kW wind turbine power coefficient using different turbulence models under the one third flow field domain which includes only one blade and the whole flow field domain which includes all the three blades.

The power coefficient curve of SAS one third domain is a little higher than that of the SAS whole flow field domain after the optimized tip speed ratio of 6. For the rest of the part, the power coefficient of the one third domain is less than that of SAS whole domain curve. The main reason is that beyond the optimized tip speed ratio, the flow is almost attached to the blade surface especially within the main power contribution range which is around $r/R=0.7$. Since in one third domain CFD simulation, the periodical boundary condition will be applied and this will result in pseudo transient flow condition. It is assumed that under the periodical

boundary conditions each of the one third domains of the whole flow fields are in the exactly same operation conditions which is not true in reflecting the real situation. Such kind of assumption will lead to the underestimation of the effect of the turbulent flow and the force acts on the wind turbine blade comparing with the whole flow field domains which can more precisely reflect the real situation than the one third domain.

For the one third domain with SST turbulence model, the value of power coefficient is almost 3% to 5% larger than that of the whole flow field domain with SAS turbulence model. The main reason is that when the unsteady turbulence model is employed, the momentum and stress equations will trigger the instability by the flow turbulence through the zone where the mesh is fine enough. Since the unsteady SST model is a RANS turbulence model which means that the high turbulent viscosity will damp the instability or in other word the instability will be averaged with time. The time average of the characteristics of the turbulence flow, especially under the intensive vortex shedding situation, will lead to the drag force underestimation as compared with the SAS turbulence model which will solve the turbulent field more accurately (Davidson and Dahlstrom 2005). These ignored drag force will cause some energy losses which could not be reflected in SST model and resulted in the calculated power coefficient a little higher than that of the SAS model.

Figure 4.14 Wind turbine rotor power coefficient versus tip speed ratio under different flow domain configuration and turbulence model

4.2.6 Torque coefficient

Figure 4.15 illustrates the torque coefficient distributions of the wind turbine rotor at different tip speed ratio under one third and the whole flow field domains conducted with different turbulence models.

Figure 4.15 shows that the present wind turbine has a relatively low torque coefficient at the beginning stage. As the tip speed ratio increase beyond 5, the torque coefficient will decline dramatically until it reaches zero. This means that the wind turbine is in no-load condition. Figure 4.15 demonstrates that the torque coefficient which obtained with SAS and SST one third domains are all higher than that of the SAS whole flow domain especially when the tip speed ratio is higher than six. The reason of this phenomenon is mainly due to the different

resolution accuracy of flow field turbulence fluctuation according to different turbulence models as discussed earlier in Section 4.2.5.

Figure 4.15 Wind turbine torque coefficient versus tip speed ratio under different flow domain configuration and turbulence model

4.2.7 Thrust coefficient

Figure 4.16 demonstrates the thrust coefficient of wind turbine rotor at different tip speed ratios. The thrust coefficient presents the force applied on the whole wind turbine rotor in the wind flow direction. The thrust force acts on the wind turbine rotor is an important parameter to the static material strength, fatigue life, blade deformation and the pressure preload blade vibration frequency. It can be seen from Figure 4.16 that the thrust coefficient increases with the increase of the tip speed ratio. Due to the present wind turbine blade, rotor design and slender blade shape, when the wind turbine at the starting stage which is corresponding to the

small tip speed ratio the wind can pass through the wind turbine rotor plane without obstruction. As the tip speed ratio keeps on increasing and approaching the no-load stage, the thrust coefficient value will reach a high level which is similar to a solid disk. The large thrust exerts on wind turbine rotor at high tip speed ratio is not only due to the drag force generated by the airfoil section, but also caused by the lift force since the direction of lift force is parallel to the incoming wind velocity.

Figure 4.16 Thrust coefficient distribution of wind turbine rotor at different tip speed ratios

4.3 Structural analysis results

4.3.1 Equivalent stress and total deformation

The equivalent stress and total deformation of the wind turbine rotor at wind speed of 10m/s are shown in Figures 4.17 and 4.18 respectively. Note that the wind turbine is rotating in anticlockwise direction in both Figures.

Figure 4.17 demonstrates that the stress induced is ranging from 289 Pa to 3.83 MPa. The highest stress induced on the blade spreads from $r/R = 0.47$ to $r/R = 0.63$ and it can also be observed that the region is relatively near the leading edge, which is the main part of the blade body where most of the power generated and the aerodynamic center where the force is acting, and is located at 0.25 chord from the leading edge (Houghton and Carpenter 2003). The high stress level will cause fatigue of the wind turbine blade and reduce its life span. Figure 4.18 illustrates that the maximum total deformation value is 4.3149×10^{-3} m. The largest total deformation occurs at the tip location. It is shown that the blade behaves like a cantilever beam with its end is fixed at the hub (Manwell et al. 2002). Therefore, when the beam is bending, the tip part will get the largest displacement from its original position. When the wind speed reaches the values of 20m/s, the stress and deformation induced do not vary much from those at the 10m/s wind speed (not shown here). This is due to that the solid blade is employed in the present study in order to simplify the physical boundary conditions at moving mesh setting.

Figure 4.17 Equivalent Stress of wind turbine rotor. Note that the rotor is rotating in anticlockwise direction and the stress unit is Pa.

Figure 4.18 Total deformation of wind turbine rotor. Note that the rotor is rotating in anticlockwise direction and the deformation unit is m.

4.3.2 Natural frequency and vibration mode

Based on the mechanical vibration, it can be shown that most of the vibration energy will concentrate on the first few modes (Chaviaropoulos et al. 2003). Therefore, in this project the first six orders of single blade vibration modes and frequencies as well as the first eighteen orders of rotor vibration modes and frequencies under pressure load will be investigated. Figures 4.19 (a) and (b) show that flap is the main type of vibration mode in first and second order respectively. There is a combination of flap and swing for the third and fourth order as shown in Figures 4.19 (c) and (d) respectively. Twist vibration is the main kind in fifth order as illustrated in Figures 4.19 (e). In sixth order the vibration mode is a combination of flap and twist as illustrated in Figures 4.19 (f). It has been shown that most of the energy during vibration will focus on first and second modes (Meirovitch 2001), therefore, the flap is the main vibration mode for a single wind turbine blade.

Figure 4.19 Vibration modes of turbine blade at wind velocity of 10m/s (a) First order, (b) Second order, (c) Third order, (d) Forth order, (e) Fifth order and (f) Sixth order

Figures 4.20 (a) and (b) show that the first and second order vibration characteristics of all the three blades are moving in an antisymmetrical way with different vibration amplitude, note that antisymmetrical movement means that among the vibration mode of the three blades, at least one blade has different vibration mode as comparing to the other two blades. Figures 4.20 (c) and (d) demonstrate that the three blades move in symmetrical way with a same vibration amplitude. Figure 4.20 (e) illustrates that two blades move in antisymmetrical way with different vibration amplitude and the third blade remains unchanged. Figures 4.20 (f) and (g) show that two blades move in antisymmetrical way with the same vibration amplitude. Figure 4.20 (h) demonstrates two blades move in antisymmetrical way with different amplitude and the third blade remains unchanged. Figure 4.20 (i) shows three blades move in a symmetrical way with the same amplitude.

It can be observed from Figure 4.20 that for three blades wind turbine rotor the vibration mode pattern can be classified into two main categories. One of them is the antisymmetrical vibration which indicates that among the three blades at least one blade has different vibration mode from the other two blades. Another category is symmetrical vibration pattern which stands for all the three blades move in the same phase at the same time. Beginning from the first order it can be observed that every three consecutive vibration orders can be treated as a cluster. Within any cluster the antisymmetrical vibration mode will show up two times and the symmetrical mode will appear only once.

Figure 4.20 Vibration modes of turbine rotor at wind velocity of 10m/s (a) First order, (b) Second order, (c) Third order, (d) Forth order, (e) Fifth order, (f) Sixth order, (g) Seventh order, (h)Eighty order and (i) Ninth order

For single blade it can be treated as a cantilever beam and for each vibration frequency there is only one corresponding vibration mode. For the whole rotor the situation is different because for the vibration frequency of the three blades forming a cluster as a whole rotor and listed in Table 4.1. Table 4.1 also illustrates that even under the same order of vibration the blade natural frequency is different from rotor vibration frequency as a whole (Malcolm 2002). Therefore, the dynamic performance of wind turbine rotor is totally different from the single blade which illustrates that the performance of single balde could not be used to represent the whole rotor performance in experiment and simulation. The difference between single blade and whole rotor shows that the whole flow field method employed in present work could deliver more reliable and accurate simulation results than those using the one third domain method which only includes one single blade in domain. For the whole wind turbine rotor the structure characteristics of each blade on the rotor is much similar to each other and the three frequencies in each cluster are close to each other. If the structure

characteristics differences among the three blades are relatively huge, then the three frequencies in each cluster will be more scattered which will result in a wide resonance range and lead to a worse rotor dynamic structure performance. Since the three blades installed on the rotor are uniform, then the close vibration frequency in each cluster can be observed which can prevent the whole rotor from working under some resonance ranges and improve the overall wind turbine performance.

As suggested by Jureczko et al. (2005) the wind speed has little effect on rotor natural frequency, the present simulation results are consistent with their conclusion. Jureczko et al. (2005) also argued that although the wind speed has little effect on vibration frequency, the angular rotation has an significant effect on frequency. The external excitation frequency has to be different from the natural frequency to avoid resonance.

Table 4.1 Vibration frequency comparison of single blade and whole rotor

Mode of Vibration	Blade Frequency (Hz)	Rotor Frequency (Hz)	
1	10.218	First Cluster	10.391
2	21.176		10.391
3	29.402		10.419
4	59.270	Second Cluster	22.557
5	93.635		23.665
6	115.810		23.666
7	162.010	Third Cluster	32.018
8	176.990		32.020
9	225.320		32.150

Figure 4.21 illustrates the single blade and wind turbine rotor under different rotating speed (in rpm) with the potential resonance points. It can be observed from Figure 4.21 that both the natural frequency of single blade and the whole rotor will increase with the increase of rotating speed. As the rotating speed increasing, the higher centrifugal force will be applied on the blade. To take the influence of rotating pre-stress caused by centrifugal force into account, an additional stiffness matrix could be added to the blade during the computational process which is named as blade stiffening effect (Qaisia and Bedoor 2005). Such increase of stiffness will lead to the increase of natural frequency for blade and rotor. Since the frequency increase is relatively small, the influence on the vibration mode of blade and rotor will be limited. A potential resonance point which indicates as a small circle in Figure 4.21 demonstrates that when the wind turbine rotating speed is approaching 214rpm, there is a higher chance to induce the structural resonance with the blade first vibration frequency. This figure provides information to the controller that the wind turbine should not operate around 214rpm, otherwise the wind turbine maybe severely damaged or even being destroyed.

Figure 4.21 Resonance diagram of a single blade and the whole wind turbine rotor with first three vibration orders

4.3.3 Blade tip displacement

Figures 4.22 (a), (b) and (c) show the displacement of blade tip in X, Y and Z direction during the wind turbine operation respectively. During the vibration period the wind turbine tip will experience the largest pressure and displacement. It can be observed in Figures 4.22 (a), (b) and (c), that there is a drastic vibration at the beginning of the fluid structure interaction. As the computational time marching on the displacement at the tip will gradually reduced and finally approach to a more stable value. The displacement range shown in Figure 4.22 (a) is the smallest because the X direction is the blade axial direction in which the force is relatively small. The Z direction shown in Figure 4.22 (c) experiences the largest

displacement range since the incoming wind flow in the same direction and inducing most of the aerodynamic force.

Figure 4.22 Displacement of tip part of blade during wind turbine operation in (a) X direction, (b) Y direction and (c) Z direction (the direction of X, Y and Z refer to coordinate system shown in Figure 4.20)

4.4 Chapter summary

In this chapter, the numerical simulation results for two-way FSI on wind turbine have been analyzed. In the flow domain, the velocity streamline and velocity vector show that when wind speed is 10m/s which is the rated wind speed, the flow is fully attached to the blade surface and there is no separation. As wind speed increases the flow separation emerges and becomes entirely detached from the upper surface of wind turbine blade. The pressure contours and coefficients of blade sections over the whole wind turbine blade under different wind speeds show that as wind speed increasing the flow separation will move towards the leading edge of the airfoil section. In the structure domain, analysis results illustrate that vibration frequency and mode of each blade in a three-blade rotor are different from each

other and it can be concluded that the simulation should prevent using the single blade to represent the whole rotor. The blade tip displacement in the Z direction experiences the largest amplitude because this direction faces the incoming flow and sustains most of the aerodynamic force, ranging from 289Pa to 3.83MPa and a maximum deformation of 4.32mm under wind speed of 10m/s. Vibration Mode analysis results show that flap is the main type of vibration in first and second modes.

CHAPTER 5 WIND TURBINE ROTOR AEROELASTIC STABILITY INVESTIGATION AND ANALYSIS

5.1 Aeroelasticity problem introduction

As an elastic object, the wind turbine blade works under the unstable aeroelasticity caused by the interaction among the aerodynamic, elastic and inertial forces during its operation. The aeroelastic stability problem is always the most important cause for the wind turbine blade structure failure. Therefore, to prevent or at least to reduce the chance of the structure fluttering is critical to maintain the long life span for the wind turbine. Generally speaking, the wind turbine will work under a wide range of wind speed. The high wind speed will cause the blade to work under deep stall condition which will increase the chance of blade flutter. Although the resonance problem of wind turbine blade and rotor have been addressed and analyzed in Section 4.3.2, it is still not able to totally eliminate the aeroelasticity problem as discussed earlier in Section 2.3.6. Another problem of using Hybrid RANS/LES model in Chapter 4 to predict the aeroelastic stability of wind turbine blade is that it requires a very long computational time for only one wind speed due to the nature of the two-way FSI. Therefore, it is necessary to develop a fast and reliable method to predict the flutter of wind turbine blade for engineering purpose. To prevent the wind turbine blade from flutter problem, the aeroelastic characteristics of wind turbine blade such as aerodynamic model, aerodynamic stiffness, aerodynamic damping and coupling of moving equations are investigated and analyzed in order to obtain a quick and reliable method to reduce or even preventing fluttering from happening.

5.2 Studying approach and model configuration

The aeroelasticity phenomenon can be classified as the static aeroelastic problem and the dynamic flutter problem. Normally, a typical airfoil section will be chosen to conduct the

analysis of the aeroelasticity problem (Riziotis et al. 2004). A typical airfoil section model is chosen (Riziotis et al. 2004) as shown in Figure 5.1.

Figure 5.1 (a) A typical airfoil section model and (b) displacement coordinates system (Dowell et al. 2005)

The flutter problem of wind turbine blade airfoil section can be presented by a two degrees freedom model demonstrated in Figure 5.1 (a) (Dowell et al. 2005). The model is supported by two springs: Spring 1 (a coil spring) is used to control the twist movement with the stiffness k and the Spring 2 (a line spring) is employed to control the vertical displacement with stiffness s . The mass center of the airfoil section is located at G, the mass of the section is denoted as m , the moment of inertia about mass center is J_0 , the twist center is located at O, the aerodynamic center is located at A, the distance between aerodynamic center (A) and mass center (G) is δ and the distance between the mass center (G) and twist center (O) is e . Figure 5.1 (b) shows the coordinate system used for the section model which presents the vertical displacement and the twist, α , angle about the zero lift line. θ is the twist angle caused by the aerodynamics force and α_0 is the initial twist angle.

5.3 Wind turbine blade airfoil section static aeroelastic problem

In static aeroelastic problem, the vertical displacement of airfoil section will be modeled as such that the displacement will not cause any additional aerodynamic force. Therefore, the vertical displacement can be ignored under such condition and the twist spring will be the only component considered in the model shown in Figure 5.1 (a). At the beginning, the airfoil section is supported by the coil spring with an initial twist angle α_0 which is corresponding to the untwist condition. When the wind turbine starts to rotate, the airfoil section will be rebalanced with a new angle of $\alpha_0 + \theta$ under the aerodynamic and spring forces. If the spring stiffness is large and the wind velocity is small, then the additional twist angle will be small. If the spring stiffness is small and the wind velocity is large, then the twist angle will be too large to cause the damage of the spring due to the over twisting of the spring limit. According to the aerodynamic force theory, the moment applied on the twist center can be written as following equations (Dowell et al. 2005):

$$M_y = M_{AC} + L(e + \delta) \quad (5.1)$$

$$L = qSC_L \quad (5.2)$$

$$M_{AC} = C_{MAC}qSc \quad (5.3)$$

$$C_L = C_{L0} + (\partial C_L / \partial \alpha)\alpha \quad (5.4)$$

$$\alpha = \alpha_0 + \theta \quad (5.5)$$

Where M_y is moment about twist center and nose up of the airfoil section is positive, M_{AC} is moment about aerodynamic center and nose up is positive, L is lift force and up is positive, $e + \delta$ is the distance between aerodynamic and twist centers, when the aerodynamic center is ahead of the twist center, it is positive by convention. Note that C_{L0} is the lift coefficient when angle of attack is equal to zero, C_{MAC} is the moment coefficient about aerodynamic

center and is a constant, $\partial C_L / \partial \alpha$ is the gradient of the lift coefficient, q is dynamic pressure and S is the area of unit length airfoil section. It is worth to high light that the C_{L0} , $\partial C_L / \partial \alpha$, and C_{MAC} are the dimensionless parameter. According to Dowell et al. (2005) in order to simplify the analysis process, the value of C_{L0} is set to zero which will not affect the results. Substitute the equations (5.2), (5.3), (5.4) and (5.5) into equation (5.1) to obtain equation (5.6).

$$M_y = (e + \delta)qS((\partial C_L / \partial \alpha)(\alpha_0 + \theta)) + qScC_{MAC} \quad (5.6)$$

If the twist spring is following the Hook's law, then moment M_y can be presented as equation (5.7)

$$M_y = K_a \theta \quad (5.7)$$

Where K_a is the twist spring stiffness.

Equate equations (5.6) and (5.7) and to have

$$K_a \theta = (e + \delta)qS((\partial C_L / \partial \alpha)(\alpha_0 + \theta)) + qScC_{MAC} \quad (5.8)$$

According to Dowell et al. (2005) the values of α_0 and C_{MAC} could set to be zero which will not affect the conclusion, then the equation could be simplified as:

$$K_a - qS(e + \delta)(\partial C_L / \partial \alpha)\theta = 0 \quad (5.9)$$

The result of equation (5.9) is a value of dynamic pressure q . Under this dynamic pressure the structure will become static unstable and the corresponding wind velocity is named as divergence speed. From the above analysis it can be found that to increase the static divergence speed the following methods can be used. One method is to move the twist center towards the aerodynamic center and the other way is to increase the stiffness of spring.

5.4 Wind turbine blade airfoil section aeroelastic stability analysis

5.4.1 Airfoil moving equations and natural frequency

Assume the coordinate of vertical displacement and twist displacement are x and θ respectively as shown in Figure 5.1. According to Bisplinghoff et al. (1996) the displacement equations of the airfoil section can be written in a matrix form as following:

$$\begin{bmatrix} J_0 & 0 \\ 0 & m \end{bmatrix} \begin{Bmatrix} \ddot{\theta} \\ \ddot{x} \end{Bmatrix} + \begin{bmatrix} k + e^2 & -es \\ -es & s \end{bmatrix} \begin{Bmatrix} \theta \\ x \end{Bmatrix} = \begin{Bmatrix} M \\ F \end{Bmatrix} \quad (5.10)$$

Where M is total moment about twist center O and F is a total force applied on a typical section.

If the influence of aerodynamic force is not considered then airfoil movement becomes a free vibration without fluid-structure interaction coupling. Then equation converts to the following form:

$$\begin{bmatrix} J_0 & 0 \\ 0 & m \end{bmatrix} \begin{Bmatrix} \ddot{\theta} \\ \ddot{x} \end{Bmatrix} + \begin{bmatrix} k + e^2 & -es \\ -es & s \end{bmatrix} \begin{Bmatrix} \theta \\ x \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix} \quad (5.11)$$

According to Bisplinghoff et al. (1996) the solution can be expressed in the following equations:

$$\begin{Bmatrix} \theta \\ x \end{Bmatrix} = \begin{Bmatrix} \Theta \\ X \end{Bmatrix} e^{\lambda_i t} \quad (5.12)$$

Insert equation (5.12) into (5.11) then the eigen value equation can be obtained as (Bottega 2006):

$$\begin{bmatrix} J_0 & 0 \\ 0 & m \end{bmatrix} \lambda^2 + \begin{bmatrix} k + e^2 & -es \\ -es & s \end{bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix} \quad (5.13)$$

The determinant of equation (5.13) is

$$\begin{vmatrix} J_0\lambda^2 + k + e^2 & -es \\ -es & m\lambda^2 + s \end{vmatrix} = 0 \quad (5.14)$$

Solving the equation (5.14) could obtain the solutions of the airfoil natural frequency as following:

$$\lambda^2 = -\frac{J_0s+m(k+e^2s)}{2mJ_0} \pm \frac{\sqrt{[J_0s+m(k+e^2s)]^2-4mJ_0sk}}{2mJ_0} \quad (5.15)$$

From equation (5.15), it can be known that if the distance between mass center and twist center is zero which means that $e = 0$, then natural frequencies of bending and twist vibration can be expressed respectively as:

$$\lambda_1 = \sqrt{s/m}, \lambda_2 = \sqrt{k/J_0} \quad (5.16)$$

The solutions of (5.16) illustrate that when the value of e equals to zero, then the bending vibration and the twist vibration is uncoupled and the wind turbine blade would not encounter the aeroelastic instability.

5.4.2 Steady state aerodynamic model

The fluid structure interaction between the flow and the airfoil can be characterized by the twist moment M and the force F . Based on the airfoil theory the lift force, L , and drag force, D , of airfoil can be shown in Figure 5.2.

Figure 5.2 Lift force and drag force on airfoil and projection direction

The lift and drag forces can be expressed in equations (2.20) and (2.21) respectively (Bisplinghoff et al. 1996).

$$L = (1/2)\rho c B C_L W^2 \quad (2.20)$$

$$D = (1/2)\rho c B C_D W^2 \quad (2.21)$$

Where ρ is the air density, c is chord, B is airfoil section length in axial direction, C_L and C_D are lift and drag coefficients respectively and W is relative wind speed.

Project the lift force and drag force into x direction then can get

$$F = L \cos \alpha + D \sin \alpha \quad (5.17)$$

$$M = F \delta = \delta (L \cos \alpha + D \sin \alpha) \quad (5.18)$$

If the flow is uniform and no perturbation, then the airfoil will not vibrate under the conditions that F and M are constants.

If the airfoil starts to vibrate due to the small perturbation and assume the vertical displacement is x and twist angle is θ . The vertical direction vibrating velocity will be $\dot{x}$ which will alter the relative wind speed. The twist angle θ will change the angle of attack of incoming flow. The relative wind speed is as follow:

$$W_0 = \sqrt{(\Omega r)^2 + V_0^2} \quad (5.19)$$

Where r is the selected radial location on wind turbine blade, Ω is the wind turbine blade angular velocity and V_0 is the incoming wind velocity.

The velocity triangle of non-vibrating airfoil and vibrating airfoil are illustrated in Figure 5.3 (a) and (b) respectively.

Figure 5.3 Airfoil section velocity triangle (a) no vibration and (b) with vibration

When the airfoil is stable without vibration, the velocity triangle is shown in Figure 5.3 (a). The airfoil should be rotating but is stable without vibration. When the airfoil is under vibrating condition the velocity triangle is illustrated in Figure 5.3 (b). The relative wind speed changes to

$$W^2 = (\Omega r)^2 + (V_0 + \dot{x})^2 \quad (5.20)$$

The angle of attack, α , becomes:

$$\alpha = \alpha_0 + \theta \quad (5.21)$$

$$\tan \alpha = (V_0 + \dot{x})/\Omega r \quad (5.22)$$

Then the following equations could be obtained:

$$dW/d\dot{x} = (V_0 + \dot{x})/W \quad (5.23)$$

$$d\alpha/d\dot{x} = \Omega r/W^2 \quad (5.24)$$

$$dW/d\theta = 0 \quad (5.25)$$

$$d\alpha/d\theta = 1 \quad (5.26)$$

Since the airfoil section vibrates near the balance position with small amplitude, then the following statement could be made:

$$\dot{x} \ll V_0 \text{ and } \dot{x} \ll W$$

Therefore, equations (5.23) and (5.24) could be rewritten as:

$$dW/d\dot{x} \approx V_0/W_0 \quad (5.27)$$

$$d\alpha/d\dot{x} = \Omega r/W^2 \approx \Omega r/((\Omega r)^2 + V_0^2) \quad (5.28)$$

According to Lagrange equation the force and moment applied on the airfoil can be written in the following format:

$$\begin{Bmatrix} M \\ F \end{Bmatrix} = - \begin{bmatrix} s_{11} & s_{12} \\ s_{21} & s_{22} \end{bmatrix} \begin{Bmatrix} \theta \\ x \end{Bmatrix} - \begin{bmatrix} d_{11} & d_{12} \\ d_{21} & d_{22} \end{bmatrix} \begin{Bmatrix} \dot{\theta} \\ \dot{x} \end{Bmatrix} \quad (5.29)$$

Where s_{ij} ($i, j = 1, 2$) represents the aerodynamic stiffness and d_{ij} ($i, j = 1, 2$) represents the aerodynamic damping.

Since the displacement x is not related to the aerodynamic force, then $s_{12} = s_{22} = 0$. Since $\dot{\theta}$ is not related to aerodynamic force, then $d_{11} = d_{21} = 0$ (Wright and Cooper 2007).

Then the aerodynamic stiffness can be obtained as following:

$$s_{11} = \partial M / \partial \theta = (\partial M / \partial W)(dW/d\theta) + (\partial M / \partial \alpha)(d\alpha/d\theta) \quad (5.30)$$

$$s_{21} = \partial F / \partial \theta = (\partial F / \partial W)(dW/d\theta) + (\partial F / \partial \alpha)(d\alpha/d\theta) \quad (5.31)$$

The aerodynamic damping can be expressed as following:

$$d_{12} = \partial M / \partial \dot{x} = (\partial M / \partial W)(dW/d\dot{x}) + (\partial M / \partial \alpha)(d\alpha/d\dot{x}) \quad (5.32)$$

$$d_{22} = \partial F / \partial \dot{x} = (\partial F / \partial W)(dW/d\dot{x}) + (\partial F / \partial \alpha)(d\alpha/d\dot{x}) \quad (5.33)$$

From equations (2.20), (2.21), (5.17) and (5.18) the following equations could be derived:

$$\partial F / \partial W = \rho c B W_0 (C_L \cos \alpha + C_D \sin \alpha) \quad (5.34)$$

$$\partial M / \partial W = \delta \rho c B W_0 (C_L \cos \alpha + C_D \sin \alpha) \quad (5.35)$$

$$\partial F / \partial \alpha = (\partial L / \partial \alpha) \cos \alpha - L \sin \alpha + (\partial D / \partial \alpha) \sin \alpha + D \cos \alpha$$

$$= 0.5 \rho c B W_0^2 \left[\frac{dC_L}{d\alpha} \cos \alpha + \frac{dC_D}{d\alpha} \sin \alpha \right] + 0.5 \rho c B W_0^2 [-C_L \sin \alpha + C_D \cos \alpha] \quad (5.36)$$

$$\partial M / \partial \alpha = \delta (\partial F / \partial \alpha) \quad (5.37)$$

$$\text{Since} \quad \partial F / \partial \alpha = \partial F / \partial \theta \quad (5.38)$$

$$\text{Then} \quad \partial M / \partial \alpha = \delta (\partial F / \partial \alpha) = \partial M / \partial \theta \quad (5.39)$$

Insert equations (5.23), (5.24), (5.25), (5.26), (5.34), (5.35), (5.36), (5.37), (5.38) and (5.39) into equations (5.30), (5.31), (5.32) and (5.33), then the corresponding linearized aerodynamic stiffness and aerodynamic damping could be obtained as follow:

$$s_{11} = 0.5 \delta \rho c B W_0^2 \left[\frac{dC_L}{d\alpha} \cos \alpha + \frac{dC_D}{d\alpha} \sin \alpha \right] + 0.5 \delta \rho c B W_0^2 [-C_L \sin \alpha + C_D \cos \alpha] \quad (5.40)$$

$$s_{21} = 0.5 \rho c B W_0^2 \left[\frac{dC_L}{d\alpha} \cos \alpha + \frac{dC_D}{d\alpha} \sin \alpha \right] + 0.5 \rho c B W_0^2 [-C_L \sin \alpha + C_D \cos \alpha] \quad (5.41)$$

$$d_{12} = \delta \rho c B W_0 (C_L \cos \alpha + C_D \sin \alpha) \frac{V_0}{W_0} + s_{11} \frac{\Omega r}{(\Omega r)^2 + V_0^2} \quad (5.42)$$

$$d_{22} = \rho c B W_0 (C_L \cos \alpha + C_D \sin \alpha) \frac{V_0}{W_0} + s_{21} \frac{\Omega r}{(\Omega r)^2 + V_0^2} \quad (5.43)$$

The typical airfoil section parameters could then be inserted into equations (5.40) to (5.43) to get the aerodynamic stiffness s_{11} , s_{21} and aerodynamic damping d_{12} , d_{22} .

Taking the aerodynamic stiffness and aerodynamic damping into consideration then the equation (5.11) becomes as follow:

$$\begin{bmatrix} J_0 & 0 \\ 0 & m \end{bmatrix} \begin{Bmatrix} \ddot{\theta} \\ \ddot{x} \end{Bmatrix} + \begin{bmatrix} 0 & d_{12} \\ 0 & d_{22} \end{bmatrix} \begin{Bmatrix} \dot{\theta} \\ \dot{x} \end{Bmatrix} + \begin{bmatrix} k + e^2 + s_{11} & -es \\ -es + s_{21} & s \end{bmatrix} \begin{Bmatrix} \theta \\ x \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix} \quad (5.44)$$

The solution obtained is shown in equation (5.12) (Bisplinghoff et al. 1996)

$$\begin{Bmatrix} \theta \\ x \end{Bmatrix} = \begin{Bmatrix} \Theta \\ X \end{Bmatrix} e^{\lambda_i t}$$

Insert equation (5.12) into (5.44) and then obtain the eigen value determinant

$$\begin{vmatrix} k + e^2s + s_{11} + \lambda^2 J_0 & d_{12}\lambda - es \\ -es + s_{21} & s + m\lambda^2 + d_{22}\lambda \end{vmatrix} = 0 \quad (5.45)$$

Then equations of eigen value can be shown as

$$r_4 \lambda^4 + r_3 \lambda^3 + r_2 \lambda^2 + r_1 \lambda + r_0 = 0 \quad (5.46)$$

$$\text{Where } r_0 = s(es_{21} + k + s_{11}) \quad (5.47)$$

$$r_1 = d_{22}(k + e^2s + s_{11}) - d_{12}(s_{21} - es) \quad (5.48)$$

$$r_2 = sJ_0 + m(k + e^2s + s_{11}) \quad (5.49)$$

$$r_3 = J_0 d_{22} \quad (5.50)$$

$$r_4 = mJ_0 \quad (5.51)$$

Assume the aerodynamic damping equals to zero, then the eigen value vector could be:

$$\lambda_{1,2}^2 = -(r_2/2r_4) \pm (\sqrt{r_2^2 - 4r_4r_0}/2r_4) \quad (5.52)$$

From the equation (5.52) it is known that when $r_0 = s(es_{21} + k + s_{11}) < 0$ which means that the aerodynamic stiffness is negative and at the same time $|es_{21} + s_{11}| > k$, then the airfoil section encounters aeroelastic instability. From the expression of aerodynamic stiffness s_{11} and s_{21} it can be shown that when the airfoil section works under stall situation, the conditions discussed above will be most possibly satisfied.

For the situation with both aerodynamic stiffness and aerodynamic damping are included, the aeroelastic stability will be judged according to the real part of eigen value. Based on the Hurwitz criterion (Bielawa 2006) the stable condition can be expressed as:

$$r_1 r_2 r_3 - r_0 r_3 - r_4 r_1^2 \geq 0 \quad (5.53)$$

5.4.3 Unsteady state aerodynamic model

The ONERA aerodynamic model consists of a set of unsteady nonlinear partial differential equations which can be used to describe the behavior of the wind turbine blade after stall condition in a wide range of angle of attack. In the ONERA model, the total circular parameters include both linear non-viscous and nonlinear viscous parts. The linear part is described by a set of first order partial differential equations which can be matched with the slender strip theory. The nonlinear terms are expressed as second order partial differential equations and the coefficient is based on the characteristics of airfoil and the transient pseudo steady lift loss caused by viscous delay effect. The drag force is relatively simple and only depends on the linear drag force. The ONERA aerodynamic stall model does not only depend on the structure movement but also rely on the damping and frequency of the flow partial differential equations (Chaviaropoulos 2001). The description of ONERA aerodynamic model equations are as follow:

$$L = \rho b(\pi b \dot{w}_0 + W(\dot{\gamma}_{1L} + \gamma_{2L})) \quad (5.54)$$

$$\ddot{\gamma}_{1L} + \frac{\lambda \dot{\gamma}_{1L}}{\tau} = 2\pi\lambda W \sin[2(\alpha - \alpha_{C_L=0})]/\tau + 2\pi\alpha_L \dot{w}_0 + d_L \dot{w}_0 \quad (5.55)$$

$$\ddot{\gamma}_{2L} + \frac{k \dot{\gamma}_{2L}}{\tau} + \frac{r \gamma_{2L}}{\tau^2} = -rW(\Delta C_L)/\tau^2 + E_L \dot{w}_0/\tau \quad (5.56)$$

$$D = \rho b[\sigma_{1D}|\Delta C_L|b\dot{w}_0 + W\gamma_{2D} + W^2 C_{DL}] \quad (5.57)$$

$$\ddot{\gamma}_{2D} + \frac{k \dot{\gamma}_{2D}}{\tau} + \frac{r \gamma_{2D}}{\tau^2} = -rW(\Delta C_D)/\tau^2 + E_D \dot{w}_0/\tau \quad (5.58)$$

Where $b = 2c$, $\tau = b/W$, $\lambda = 0.17$, $\alpha_L = 0.53$, γ_{1L} and γ_{2L} are the lift linear and nonlinear terms, γ_{2D} is a drag term, w_0 is velocity component perpendicular to the chord, α is angle of attack, $\alpha_{C_L=0}$ is zero lift angle of attack, C_{DL} is minimum value of drag polar, ΔC_L and ΔC_D are the variation from 2-D lift and drag polars caused by viscous effects respectively, $d_L = -0.075|\Delta C_L|$, $E_i = E_{2i}(\Delta C_L)^2$ where $i = L, D$, $\sqrt{r} = 0.25 + 0.25(\Delta C_L)^2$, $k = 0.25 + 0.30(\Delta C_L)^2$, $E_L = -18/\pi$, $E_D = -4.5/\pi$ and $\sigma_{1D} = -4.5/\pi$.

The derivation of airfoil movement equations is similar to that of steady aerodynamic model. Since the ONERA model includes the nonlinear viscous term, the corresponding parameter used in this equation required to include the entire dynamic phenomenon such as deep stall which will normally require the usage of LES turbulence model other than RANS turbulence model. Therefore, to have a good balance between precisely capture airfoil characteristics such as lift and drag coefficients with dynamic stall phenomenon and computational time, the hybrid RANS/LES turbulence model which has been employed in Chapter 4 will be adopted. The hybrid RANS/LES turbulence model has shown its ability in predicting the wind turbine blade aerodynamic performance under stall working condition which has been verified with NREL phase VI experimental data. This model will be used to conduct the calculation in the present study. This method will help wind turbine engineers to save a lot of time by allowing them to use the ONERA aerodynamic model instead of whole flow field CFD simulation. The wind industrial engineers could use the normal desktop computer other than hyper computer with a relative fast and economical way to conduct the assessment of aeroelastic stability analysis with yet reliable results.

5.4.4 Determination of airfoil section parameters

The coordinate system will be set as following: the point of chord line intersects with the leading edge will be defined as origin, the X and Y coordinate are indicated in Figure 5.4.

Figure 5.4 Airfoil coordinate systems at leading edge and mass center

5.4.4.1 Area of airfoil section

$$A = \iint_D dx dy = \int_0^c dy \int_{\phi_1(y)}^{\phi_2(y)} dx \quad (5.59)$$

Where c is the chord of airfoil section, $\phi_1(y)$ and $\phi_2(y)$ are upper and lower surfaces respectively and are function of y coordinates.

5.4.4.2 Mass center of airfoil section

Assume the wind turbine blade material is uniform and density is constant. Then the coordinate of mass center (G) could be obtained as follow:

$$\bar{X} = \frac{1}{A} \iint_D x dx dy \quad (5.60)$$

$$\bar{Y} = \frac{1}{A} \iint_D y dx dy \quad (5.61)$$

5.4.4.3 Polar inertia moment to mass center

If coordinate point of origin is O then the polar inertia moment is:

$$I_x = \int_D y^2 dx dy \quad (5.62)$$

$$I_y = \int_D x^2 dx dy \quad (5.63)$$

If coordinate point of origin is G then the polar inertia moment is:

$$I_{\eta} = I_y - A\bar{X}^2 \quad (5.64)$$

$$I_{\xi} = I_x - A\bar{Y}^2 \quad (5.65)$$

Then the polar moment of inertia to mass center will be:

$$I_p = I_{\eta} + I_{\xi} \quad (5.66)$$

5.4.4.4 Moment inertia of airfoil section

$$J_0 = \iiint_{\Omega} (\eta^2 + \xi^2) \rho d\eta d\xi dz = \rho \int_0^B [\iint_D \eta^2 + \xi^2) d\eta d\xi] dz = \rho I_p B \quad (5.67)$$

Where z axis is passing through the mass center and perpendicular to airfoil section in the radial direction, B is the thickness of the airfoil section.

5.4.4.5 Stiffness of vertical and twist spring

$$k = GI_p/r \quad (5.68)$$

$$s = 3EI_{\eta}/r^3 \quad (5.69)$$

Where G and E are the shear modulus and elastic modulus of the wind turbine material respectively, r is the radial distance from the hub center to the airfoil section.

5.4.4.6 δ and e

$$\delta = \bar{Y} - c/4 \quad (5.70)$$

$$e = Y_0 - \bar{Y} \quad (5.71)$$

Where c is the chord and Y_0 is the coordinate of twist center. To simplify the calculation, it is assumed that aerodynamic center is located at the one quarter of the chord from the leading edge.

5.4.5 Aerodynamic characteristics of airfoil section

A typical airfoil section of S822 which is spread from tip to $r/R=0.3$ of the wind turbine blade is chosen for the study of aerodynamic parameter. The material used is fiber glass and the

wind speed is 10 m/s. According to equations (5.59) to (5.71) the airfoil S822 coordinate data will be used to calculate the real part of eigen value in vertical and twist direction of airfoil section employed in the vibration equation and the results obtained are shown in Figures 5.5 and 5.6 respectively of tip speed ratio at $r/R=0.5$

Figure 5.5 shows that when the tip speed ratio is higher than 1.7 the real part is negative value. As the reduction of tip speed ratio, the real part will switch from negative to positive. The phenomenon is due to the augment of the angle of attack when the corresponding tip speed ratio keeps reduction. With the small angle of attack, the vibration of vertical displacement trends to converge. On the other hand, when the angle of attack is large the airfoil will be in stall condition and the vertical displacement vibration trends to diverge. Figure 5.6 demonstrates the real part of eigen value of twist vibration is very small and the displacement in this direction is converging.

Figure 5.5 The real part of the eigen value of the airfoil in vertical direction versus tip speed ratio at $r/R=0.5$

Figure 5.6 The real part of the eigen value of the airfoil in twist direction versus tip speed ratio at $r/R=0.5$

Figures 5.7 (a) and (b) show the comparison of present computed lift and drag coefficients with experimental data (Selig and Granahan 2004) using SAS-SST method and airfoil S822. In Figure 5.7 (a) the present CFD lift coefficient results match the experimental data very well for the angle of attack varying from -10° to 6° and shows slight discrepancy from 6° to 14° . In Figure 5.7 (b) the present CFD data of drag coefficient is very close to experimental data for the angle of attack varying from -10° to 6° . Although beyond that the discrepancy between the two sets of data is large, the absolute value of difference is still very small (Note that the scale used in vertical axis in Figure 5.7 (b) is much smaller than that in Figure 5.7 (a)) which indicates the accuracy of the present SAS-SST turbulence model. The results show that the present turbulence model used could be employed to calculate the lift and drag coefficients of airfoil S822 under different angle of attack and will be used further in the subsequent aeroelastic stability analysis.

Figure 5.7 Lift and drag coefficient comparison between experimental data and CFD data of airfoil S822 (a) lift and (b) drag

The experimental data is available only within the range of angle of attack varying from -9° to 14° which is not big enough for the aeroelastic investigation purpose. Therefore, the SAS-SST hybrid turbulence model is employed to obtain the lift and drag coefficients for angle of attack varying from -9° to 30° and then use the least square fitting method to generate the extension of lift and drag coefficient curves which are shown in Figures 5.8 (a) and (b) respectively. It can be observed from Figure 5.8 (a) that the lift coefficient will offer maximum value when the angle of attack reaches around 13° and then begin to drop down. From 13° to 17° the reducing of the lift coefficient is very drastic and then become very gentle after 17° . Figure 5.8 (b) shows that the drag coefficient increase very gently when angle of attack is less than 13° and then the increasing of the drag coefficient become very drastic when the angle of attack is large than 13° .

Figure 5.8 Angle of attack range extension of lift and drag coefficient

5.5 Aeroelastic stability assessment and parameter analysis

5.5.1 Introduction of aeroelastic calculation

Based on the vibration equation and the aerodynamic model which have been introduced and derived in the early chapters, the movement of the airfoil section will be analyzed using the numerical FSI method to obtain the bending and twisting displacement pattern. The airfoil section movement model will adopt the configuration in Figure 5.1 (a) and (b) and the aerodynamic model will be chosen as steady model and NOERA unsteady model respectively. The wind turbine blade section will be selected at location of $r/R=0.75$ and the corresponding airfoil S822 will be used. All the other parameters will remain the same as mentioned in previous paragraphs. It is assumed that at the initial stage the wind turbine blade is in the static balance condition and then the external excitation cause the blade to start vibrating. The behavior of the vibration will be analyzed to find out whether its amplitude will be converged or diverged under different conditions.

5.5.2 Mathematical approach

For the wind turbine working under certain operating conditions with corresponding tip speed ratio, wind velocity and airfoil section chord at specified location, the aerodynamic stiffness and aerodynamic damping can be determined. Assuming that at time $t_0 = 0$, there is some perturbation and one of the x_0 and θ_0 is not equal to zero, then the initial aerodynamic force and aerodynamic moment are described as F_0 and M_0 which are constants. Insert the values of F_0 and M_0 into equation (5.10) to calculate the value of x_1 and θ_1 at time $t_1 = t_0 + \Delta t$. When the values of x_1 , $\dot{x}_1$, θ_1 and $\dot{\theta}_1$ are obtained and substitute into equation (5.29) to calculate F_1 and M_1 , then substitute the found F_1 and M_1 into equation (5.10) again to calculate next x_2 and θ_2 . The iteration will be carried out to get the aerodynamic force, displacement, velocity and acceleration. The fourth order of Runge-Kutta numerical method is employed and the program is written as follow:

Assume $p_1 = \theta$, $p_2 = \dot{\theta}$, $p_3 = x$, and $p_4 = \dot{x}$ then the following equations could be obtained.

$$p'_1 = p_2 \quad (5.72)$$

$$p'_2 = (M - (k + e^2 s)p_1 + esp_3)/J_0 \quad (5.73)$$

$$p'_3 = p_4 \quad (5.74)$$

$$p'_4 = (F + esp_1 - sp_3)/m \quad (5.75)$$

Then

$$p_{1,n+1} = p_{1,n} + \frac{1}{6}(k_1 + 2k_2 + 2k_3 + k_4) \quad (5.76)$$

$$p_{2,n+1} = p_{2,n} + \frac{1}{6}(l_1 + 2l_2 + 2l_3 + l_4) \quad (5.77)$$

$$p_{3,n+1} = p_{3,n} + \frac{1}{6}(m_1 + 2m_2 + 2m_3 + m_4) \quad (5.78)$$

$$p_{4,n+1} = p_{4,n} + \frac{1}{6}(q_1 + 2q_2 + 2q_3 + q_4) \quad (5.79)$$

The expressions of k , l , m and q are as follow:

$$\begin{cases} k_1 = hp_{2,n} \\ l_1 = \frac{h}{J_0}(M_n - (k + e^2s)p_{1,n} + esp_{3,n}) \\ m_1 = hp_{4,n} \\ q_1 = (F_n + esp_{1,n} - sp_{3,n}) \end{cases} \quad (5.80)$$

$$\begin{cases} k_2 = h(p_{2,n} + l_1/2) \\ l_2 = \frac{h}{J_0}(M_n - (k + e^2s)\left(p_{1,n} + \frac{k_1}{2}\right) + es(p_{3,n} + m_1/2)) \\ m_2 = h(p_{4,n} + q_1/2) \\ q_2 = (F_n + es(p_{1,n} + \frac{k_1}{2}) - s(p_{3,n} + m_1/2)) \end{cases} \quad (5.81)$$

$$\begin{cases} k_3 = h(p_{2,n} + l_2/2) \\ l_3 = \frac{h}{J_0}(M_n - (k + e^2s)\left(p_{1,n} + \frac{k_2}{2}\right) + es(p_{3,n} + m_2/2)) \\ m_3 = h(p_{4,n} + q_2/2) \\ q_3 = (F_n + es(p_{1,n} + \frac{k_2}{2}) - s(p_{3,n} + m_2/2)) \end{cases} \quad (5.82)$$

$$\begin{cases} k_4 = h(p_{2,n} + l_3) \\ l_4 = \frac{h}{J_0}(M_n - (k + e^2s)(p_{1,n} + k_3) + es(p_{3,n} + m_3)) \\ m_4 = h(p_{4,n} + q_3) \\ q_4 = (F_n + es(p_{1,n} + k_3) - s(p_{3,n} + m_3)) \end{cases} \quad (5.83)$$

In each iteration, time interval should be selected such that it is less than the natural frequency of the airfoil system to ensure the stability of the numerical iteration.

5.5.3 Cases study and results analysis

The regular of airfoil section movement in vertical and twist directions as well as the development trend of vibration frequency and phase plane will be discussed and analyzed in this section under several different wind turbine working conditions.

5.5.3.1 Blade aeroelastic stability when $\lambda=3$

Figures 5.9 (a) and (b) show the bending and torsion displacement of blade section when $\lambda=3$. The displacements of both bending and torsion are relative small and converge quickly after initial external excitation. Figures 5.9 (c) and (d) demonstrate the bending and torsion velocity pattern are decaying fast and similar to those of displacement. Figures 5.9 (e) and (f) illustrate the trend of bending and torsion acceleration respectively. The amplitude of acceleration will quickly converge to zero after 10 seconds of initial perturbation. Figures 5.9 (g) and (h) show the phase plane of bending and torsion movement. From these figures it can be observed that all of the curves tend to converge to zero which proves that the bending and torsion vibration are stable.

Figure 5.9 Airfoil section aeroelastic stability analysis when tip speed ratio, $\lambda=3$ (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane

5.5.3.2 Blade aeroelastic stability when $\lambda=2.4$

Figures 5.10 (a) to (h) show the bending and torsion displacements, velocity, acceleration and phase plane respectively at $\lambda=2.4$. The curve pattern and development trend of each parameter are similar to the corresponding part in Figure 5.9. The only difference is that the amplitudes of vibration in both bending and torsion direction at $\lambda=2.4$ are a little bit higher than those in Figure 5.9. Overall the trajectory of bending and torsion movement in phase plane illustrates that under tip speed ratio of 2.4 the wind turbine blade is still in aeroelastic stable condition.

(a)

(b)

(c)

(d)

(e)

(f)

Figure 5.10 Airfoil section aeroelastic stability analysis when tip speed ratio, $\lambda=2.4$ (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane

5.5.3.3 Blade aeroelastic stability when $\lambda=1.7$

Figure 5.11 demonstrates the movement characteristics of blade airfoil section under tip speed ratio of 1.7. Figures 5.11 (a) and (b) show that the vibration displacement in bending and torsion direction keeps increasing from the beginning of external excitation being applied and the amplitude increases to infinity in a short time. Figures 5.11 (c) to (f) illustrate the same trend of vibration velocity and acceleration in both bending and torsion. Figures 5.11 (g) and (h) show that movement trajectory in phase plane for both bending and torsion directions are diverging. The bending phase trajectory starts from the origin and move towards infinity in a circular shape in clockwise direction as demonstrated in Figure 5.11 (g). The torsion phase plane trajectory starts from the point which is the initial twist angle of the airfoil section of the wind turbine blade and also move towards infinity in a circular shape in clockwise direction as indicated in Figure 5.11 (h). Figure 5.11 shows that the wind turbine

blade section experienced aeroelastic instability under the tip speed ratio of 1.7. For the present study of different λ values, it is shown that where $\lambda \leq 1.7$ the wind turbine blade will encounter aeroelastic instability and the system is stable when λ is higher than 1.7. It is clear that the system should not be operated with λ less than 1.7 which is the threshold of instability.

(a)

(b)

(c)

(d)

(e)

(f)

Figure 5.11 Airfoil section aeroelastic stability analysis when tip speed ratio, $\lambda=1.7$ (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane

5.5.3.4 Blade aeroelastic stability when $\lambda=1.5$

From Figures 5.12 (a) to (f), it can be observed that the vibration in both bending and torsion directions are diverging due to the aerodynamic energy keeps injecting into the wind turbine for $\lambda=1.5$. The energy absorption is due to the coupling of the bending and the torsion motion of the wind turbine which forms a feedback loop to enhance the airfoil vibration. The diverging process of the bending and torsion movement is much faster than the divergent case when $\lambda=1.7$. The main reason for this faster displacement, velocity and acceleration divergence than those at $\lambda=1.7$ is because when the tip speed ratio decreases to 1.5 the aerodynamic damping reduces tremendously which lead to the serious aeroelastic instability. From the comparison of Figures 5.9 to 5.12 it can be observed that as the tip speed ratio decreases or the wind speed increases the vibration amplitude of airfoil section will increase and finally result in flutter situation. When $\lambda=1.7$ the vibration of airfoil has worked under

flutter condition which means that the working wind speed for the designed 5kW wind turbine should not be over 35m/s with fixed rotating speed.

(a)

(b)

(c)

(d)

(e)

(f)

Figure 5.12 Airfoil section aeroelastic stability analysis when tip speed ratio, $\lambda=1.5$ (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane

5.5.3.5 Phenomenon of frequency overlap

Figure 5.13 illustrates the vibration frequency for the bending and torsion with the wind speeds. It can be observed that the two vibration frequencies come closed to each other, and finally the coupling of bending and torsion vibrations may occur. The coupling of bending and torsion vibrations will absorb energy from the wind and come to the critical value of flutter. The curves shown in Figure 5.13 indicate that when the wind speed increase to a critical value, the frequency of the bending and torsion will be the same and result in flutter. Due to the characteristics of unsteady aerodynamic model the two frequencies will approach each other at around 30Hz but will not converge to reach a same value finally.

Figure 5.13 Bending and torsion vibration frequency move towards overlap as the increasement of wind speed

5.5.3.6 Influence of mass center location on aeroelastic stability

Figure 5.14 shows the variation of reduced wind speed $2V/c\omega_t$ along the mass center location, where V is the corresponding flutter wind speed, c is the chord, ω_t is the torsion vibration frequency and the parameter x represents the mass center location on the chord of the airfoil section. Figure 5.14 shows that for the same mass center location the flutter wind speed calculated based on the steady aerodynamic model is more conservative as comparing with that from the ONERA unsteady aerodynamic model. The reason is because the steady model assumes that the flutter is only related to the additional aerodynamic force caused by

vibration and ignores the wash down acceleration as well as the reduced frequency which represents the impact of interaction between the different points on the airfoil section.

Figure 5.14 Impact of mass center location on the flutter wind speed

Figure 5.14 also illustrates that when the mass center is beyond the twist center ($x=0.35c$), then the critical flutter wind speed will become infinity which means that the flutter will not happen. When the mass center is located after the twist center, then the corresponding critical flutter wind speed will keep decreasing as mass center moves towards the trailing edge of airfoil section. When the mass center is located between the $0.4c$ to $0.6c$ the critical flutter wind speed drops very dramatically and the decreasing speed has become gentle after the location of $0.6c$. The reason for this phenomenon is due to the shifting of the mass center backwards which will increase the rotational inertia of torsion movement. The increase of the

rotational inertia will reduce the torsion frequency and shift more close to bending frequency and enhance the coupling between the bending movement and the torsion movement. Figure 5.14 demonstrates that it is more accurate to employ the ONERA unsteady aerodynamic model than the quasi-steady aerodynamic model in predicting the wind turbine aeroelastic stability. The figure also implies that one approach to optimize the wind turbine FSI performance is to shift the mass center close to the twist center to increase the blade flutter wind speed.

5.5.3.7 Impact of different wind turbine materials on aeroelastic stability

Figures 5.15 (a), (b) and (c) show the real part of eigen values in bending direction for three different wind turbine materials under different tip speed ratios. The real part amplitude value of fiber glass is the biggest among the three selected materials and the steel poses the smallest real part amplitude value. Since the negative real part indicates the stable movement and the positive real part implies the unstable vibration, it could be concluded that the blade made of fiber glass is more likely to encounter flutter problem than the steel blade. These results mean that the blade with lower stiffness will get higher chance to experience aeroelastic instability and the divergent vibration will develop within a short time. Therefore, a high stiffness of the blade structure will enhance the aerodynamic stability and provide another option in improving the wind turbine blade FSI performance.

Figure 5.15 Real part of eigen values in bending direction of three differential wind turbine blade materials (a) fiber glass, (b) aluminum alloy and (c) steel under differential tip speed ratios

5.6 Chapter summary

In this chapter, the wind turbine rotor aeroelastic stability has been studied and analyzed. The aeroelasticity problem, studying approach and model configuration are introduced respectively. The analysis results illustrate that the wind turbine will be stable when the wind speed is below 35m/s and will experience flutter when wind speed is over 35m/s. The

phenomenon of frequency overlap has been observed in the analyzed result which is the main reason for wind turbine blade fluttering. The bending and torsion frequencies are 23.94Hz and 38.46 Hz respectively under TSR 3 and both move towards 30Hz under TSR 1.7. When TSR is smaller than 1.7 the wind turbine will experience flutter due to the overlapping of bending and torsion frequencies. The influence of mass center and different materials shows that moving the mass center towards the twist center of airfoil as well as increasing the blade material stiffness will increase the critical flutter wind speed.

CHAPTER 6 MULTIPLE OBJECTIVES OPTIMIZATION OF WIND TURBINE PERFORMANCE

6.1 Purpose of optimization

The rotor which is the essential part of the wind turbine in converting the wind energy into mechanical energy. In this conversion process the aerodynamic shape of the blade is a key factor which could directly affect the power efficiency of wind turbine. Therefore, it is necessary to select a wind turbine blade with an excellent aerodynamic shape to achieve the maximum power efficiency. The traditional designing method takes the maximum power efficiency of each element on wind turbine blade as the design target. When using this traditional designing method, some important factors such as the interaction of each element and the different wind speeds are ignored which will result in the low efficiency. To overcome the disadvantage of the traditional designing method, an improved method with the target of maximizing annual power output will be developed. The improved method will use the chord and twist angle of each blade element as designing parameters and the genetic algorithm to conduct the optimization designing process.

6.2 Multiple objectives optimization

In the wind turbine rotor optimization process, there are a lot of parameters to consider, they may not be compatible to each other and even contradictory under most of the operation conditions. In the wind turbine designing process, the following main factors will be considered in the order of priority.

- (1) The wind turbine power efficiency is maximized
- (2) Long life span under varying aerodynamic force
- (3) Blade vibration is relatively small

(4) High flutter wind speed

(5) Blade material cost is low

(6) Blade structure is stable during operation

The wind turbine optimization designing process needs to take a balance among all these factors which becomes a multiple objectives optimization problem.

The multiple objectives optimization is defined as searching for the optimized values of the designing parameters within the acceptable range under the given constraints to achieve the designing objective, i.e., to reach the best performance. Normally in multiple objectives optimization it is very difficult to ensure all these objectives to become optimal. In the solving process the optimal value of one sub objective will always lead to one or several other sub objectives being less satisfied and make the optimization process become contradictory which is the characteristics of the multiple objectives optimization problem. Another feature of the multiple objectives optimization problem is that there is no uniform measuring standards for the optimal value of all the sub objectives which also increase the difficulty in judging the effectiveness of the method. The multiple objectives optimization problem could be described as follow:

$$U - \min F(x) = [f_1(x), f_2(x), \dots, f_m(x)]^T \quad x \in R^n \quad (6.1)$$

$$\text{Subject to } \begin{cases} g_u(x) \geq 0, u = 1, 2, \dots, p, p < n \\ h_v(x) = 0, v = 1, 2, \dots, q, q < n \end{cases}$$

Where m is the number of target function, n is the number of designing parameter, p and q are the unequal and equal constraints number respectively. $U - \min F(x)$ means for the minimum value of all the sub function in the target vector of $F(x) = [f_1(x), f_2(x), \dots, f_m(x)]^T$. Since the multiple objectives optimization is different

from single objective problem, therefore, the definition of the optimal solution of the target vector function is as follow:

Assume $x \in R^m$ is a set constrains for multiple objectives problem, $F(x) \in R^n$ is the target vector function. If $x' \in X$ and $F'(x) \leq F(x)$, $\forall x \in X$, then x' is the absolute optimal value of the multiple objectives problem. The absolute optimal value only exists when each sub function has optimal value under the same solution which is a very special situation. Therefore, the absolute optimal value of multiple objectives problem is not always available. In the practical engineering designing process, a set of solution will be obtained and some further methods are employed to pick one or several solutions as the optimal solution of the problem (Simon 2013).

6.3 Genetic algorithm

Genetic algorithm is a kind of random searching algorithm which is based on the number of the population to simulate the natural choice and evolution. This method uses the creature evolution theory and program to convert the designing parameter into the individual of the population to form the chromosome. The gene of the chromosome will keep the character of individual and will be given some adaptive value which is a benchmark to measure the individual based on evaluation. Some selection strategies will be applied to the father generation to select the new individual to form a new generation according to the adaptive value and the chromosome with a good adaptive value will be given more opportunity to reproduce. Based on the method of duplication, hybrid, mutant and assessment, the new generation will be evaluated continuously and the iteration will gradually lead to the optimal zone in the search area until an optimal solution is found. Although the randomness of this method is mainly about the mutant randomness, the genetic algorithm uses the natural choice mechanism to achieve the specified direction of the evolution process. Since the genetic

algorithm is a suitable method for nonlinear discontinuous function with multiple peak and has global optimization stability for the aimed function as well as the multiple search approach and high convergency, eventually this method will be used to conduct the multiple objectives optimization of wind turbine blade (Sivanandam and Deepa 2008).

6.4 Wind turbine blade optimization

The aerodynamic behavior of wind turbine blade will have great influence on the integrated wind turbine performance. Another optimization target should be taken into account is the load applied on wind turbine rotor. The multidimensional optimization of small wind turbine now will include the noise, starting performance, blade structural behavior and mass which have been discussed by (Sessarego and Wood 2015) and (Pourrajabian et al. 2016) in details. Although the noise and starting performance are important for small wind turbine optimization, these two factors are not being considered in the present study due to the financial constrained in carrying out the corresponding experiment to verify the optimization results. Taking all factors into consideration the multiple objectives optimization in present study will include the maximizing aerodynamic performance and minimizing blade loading to provide useful guide for practical wind turbine engineering design.

6.4.1 Define target function

6.4.1.1 Aerodynamic performance function

For fixed pitch wind turbine the basic benchmark is the power output annually which is equal to the annual average power multiply the operating hour. Therefore, the 5kW wind turbine power efficiency with parameters of rotor diameter, rotating speed, chord and twist angle discussed in Chapter 3 under rated wind speed will be adopted as the designing target of aerodynamic performance function.

$$f_1(x) = f_1(x_1, x_2, \dots, x_n; c_1, c_2, \dots, c_n; \theta_1, \theta_2, \dots, \theta_n; \delta_1, \delta_2, \dots, \delta_n) \quad (6.2)$$

$x_i (i = 1, 2, \dots, n)$ the distance between i th section and blade root

$c_i (i = 1, 2, \dots, n)$ the chord of i th section

$\theta_i (i = 1, 2, \dots, n)$ the twist angle of i th section

$\delta_i (i = 1, 2, \dots, n)$ the length of i th section

6.4.1.2 Blade loading function

The force loading applied on wind turbine blade during operation could be classified as inertial loading, aerodynamic loading and gravity loading. The loading coordinate system is shown in Figure 4.21 (Page 129). The aerodynamic force will induce blade bending and torsion. The centrifugal force will cause blade stretching, bending and torsion. The gravity force will cause blade stretching, compressing, bending and torsion. The blade loading function is designed as follow:

$$f_2(x) = \|q\| = \|(q_a, q_g, q_i)\| \quad (6.3)$$

where q_a is aerodynamic force per unit length, q_g is gravity force per unit length and q_i is inertial force per unit length.

6.4.1.3 Optimization model function

The target function is the wind turbine power efficiency and blade loading. The evaluation function will be constructed according to the characteristics of wind turbine blade problem shown in equation (6.4).

$$F(x) = f(f_1(x), f_2(x)) \quad (6.4)$$

Where $F(x)$ is the total target function.

6.4.1.4 Constrains

The optimization function will be solved under the following constrains:

$$\begin{cases} x_i = x_{i-1} + (x_n - x_1)/n \\ c_{max} \geq c_1 \geq c_2 \cdots \geq c_i \geq \cdots \geq c_n \geq c_{min} \\ \theta_{max} \geq \theta_1 \geq \theta_2 \cdots \geq \theta_i \geq \cdots \geq \theta_n \geq \theta_{min} \\ \delta_{max} \geq \delta_1 \geq \delta_2 \cdots \geq \delta_i \geq \cdots \geq \delta_n \geq \delta_{min} \end{cases} \quad (6.5)$$

Where c_{max} and c_{min} are the maximum and minimum chord of optimized section respectively, θ_{max} and θ_{min} are the maximum and minimum twist angle of optimized section respectively, δ_{max} and δ_{min} are the maximum and minimum section length of optimized section respectively.

6.4.1.5 Optimization flow chart

The optimization program flow chart is shown in Figure 6.1

Figure 6.1 Flow chart of optimization program with genetic algorithm

When the individual adaptability function is calculated, the value of the aerodynamic force and the loading will be necessary. The calculation method and accuracy of the target function will affect the design result. Therefore, the strip theory will be employed to calculate aerodynamic force with the consideration of tip and hub losses and the value of the total loading on wind turbine blade.

6.4.2 Optimization results and discussion

Figures 6.2 (a) and (b) show the comparison of chord and twist angle of initial design and optimized design respectively.

It can be observed in Figure 6.2 (a) that the overall chord based on optimized design is shorter than the initial design and the chord transition between different blade elements is more gentle and smoother than the initial design. The chord, especially the portion near the tip, is much shorter than the initial design which will not only cut down the production cost but also will alleviate the total force loading applied on the blade to prolong the wind turbine life span. Figure 6.2 (b) demonstrates the twist angle of wind turbine blade according to initial design and optimized design. The optimized design of twist angle are larger than the corresponding value of initial design. The difference between the twist angle of initial design and optimal design is tiny near the blade root part and become relatively large between middle to tip region. The increasing of twist angle will fortify the wind turbine performance in stall conditions as well as reducing the aerodynamic loading on wind turbine blade to reduce the chance of structure fatigue.

Figure 6.2 Comparison of chord and twist angle between initial design and optimized design (a) chord and (b) twist angle

Figures 6.3 (a), (b) and (c) illustrate the power coefficient, torque coefficient and thrust coefficient of wind turbine blade based on initial design and optimized design.

It can be observed from Figure 6.3 (a) that the power coefficient of optimized blade is larger than that of the initial design when tip speed ratio is less than 6 due to the increase of twist angle in the optimized design. The maximum value of power coefficient appears near around tip speed ratio 6.2 which indicates the improvement of the wind turbine power performance at high wind speed and the maximum value of power coefficient is higher than the initial design which subsequently increase the power output of wind turbine. The torque coefficient curve as shown in Figure 6.3 (b) indicates that there is an obvious improvement of torque performance at high wind speed which is mainly caused by the increasement of twist angle. Although the torque coefficient is less than the initial design when tip speed ratio is higher than 6.2, the difference is relatively small and would not impose significant effect on the wind turbine overall performance. From the relationship of thrust coefficient curve and tip speed ratio shown in Figure 6.3 (c) it can be observed that the thrust force applied on optimized designing wind turbine blade is smaller than that of initial designing which will not only reduce the total loading on the wind turbine rotor but also will prolong the wind turbine life cycle.

Figure 6.3 Comparison of wind turbine performance between initial design and optimized design (a) power coefficient, (b) torque coefficient and (c) thrust coefficient

Figure 6.4 demonstrates the vibration characteristics of blade airfoil section at the location of $r/R=0.75$ under tip speed ratio of 1.7 based on the optimized design.

Figures 6.4 (a) to (f) show the aeroelastic characteristics of wind turbine blade airfoil section under tip speed ratio equals to 1.7 with optimized design and Figures 6.4 (g) and (h) illustrate the phase plane of bending and torsion components respectively under the same situation. From these figures it could be observed that the blade is under aeroelastic stability when $\lambda=1.7$. Although the oscillating amplitude of displacement, velocity and acceleration are high at the initial stage, the vibration amplitude reduces slowly to zero in a relatively gentle process. In the initial design (please refer to Figure 5.11) the wind turbine blade will experience flutter under this tip speed ratio (1.7) which implies that wind turbine could not work at and lower than the corresponding wind speed. In the optimized design the wind

turbine flutter wind speed has been improved to a higher value to extend the wind turbine working range. These results show that the optimization design will not only improve the wind turbine power and loading performance but also will provide an effective approach to delay the wind turbine flutter with higher wind speed.

Figure 6.4 Airfoil section aeroelastic stability analysis when $\lambda=1.7$ with optimized blade design (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane

Figures 6.5 (a) to (f) show the aeroelastic characteristics of wind turbine blade airfoil section under tip speed ratio equals to 1.6 with optimized design and Figures 6.4 (g) and (h) illustrate the phase plane of bending and torsion components respectively under the same situation. From these figures it could be observed that the blade is under aeroelastic instability when $\lambda=1.6$. Although the blade experience instability under $\lambda=1.6$, the diverge amplitude of displacement, velocity, acceleration are relatively gentle and slow which indicate that $\lambda=1.6$ is the initial flutter speed of the optimized design wind turbine blade.

Figure 6.5 Airfoil section aeroelastic stability analysis when $\lambda=1.6$ with optimized blade design (a) bending displacement, (b) torsion displacement, (c) bending velocity, (d) torsion velocity (e) bending acceleration, (f) torsion acceleration, (g) bending phase plane and (h) torsion phase plane

6.5 Chapter summary

In this chapter, the multiple objectives optimization of wind turbine performance including the purpose of optimization, multiple objectives optimization method and genetic algorithm are presented. The target function which includes aerodynamic performance function, blade loading function and optimization model function has been illustrated. The wind turbine blade optimization results have shown that the design parameters of chord, twist angle, power coefficients, torque coefficients and thrust coefficients are being improved to attain better performance, even the threshold of tip speed ratio is being reduced from 1.7 to 1.6.

CHAPTER 7 CONCLUSIONS AND FUTURE WORKS

7.1 Conclusions

A new computational method on a whole flow field two way wind turbine fluid structure interaction has been developed and presented. The new approach has shown to have satisfactory and encouraging results in improving wind turbine design and performance, the conclusion are discussed as following.

1. The parameters of 5kW wind turbine are obtained based on the classical BEM method and the 3D model of wind turbine rotor is generated in SolidWorks. The hybrid turbulence model used in the present work has been verified according to the NREL phase VI experimental data and has been shown to be a reliable turbulence model for conducting wind turbine numerical simulation. The detailed simulation configuration of the whole flow field two-way FSI investigation of flow domain and structure domain have been introduced and conducted respectively. In flow and structure domains, the mesh independency is being tested and analyzed in terms of the size of domain and the mesh characteristics. The numerical simulation parameters such as the converging criteria, time step and total simulation are also introduced and tested.

2. The CFD simulation results including the velocity streamline and velocity vector show that at the rated wind speed of 10m/s, the flow is fully attached to the blade surface and there is no separation. As wind speed increases the flow separation emerges and becomes entirely detached from the upper surface of wind turbine blade. The pressure contours and coefficients of blade sections over the whole wind turbine blade under different wind speeds show that as wind speed increasing the flow separation will move towards the leading edge of the airfoil section due to the high angle of attack. The flow separation near the root part is not prominent for all the wind speeds due to the stall delay effect caused by the rotating of the

wind turbine blade. The comparisons of wind turbine power and torque coefficients with different CFD methods demonstrate that whole flow field method with hybrid turbulence model could deliver the most reliable and accurate simulation results. Structure domain results illustrate that vibration frequency and mode of each blade in a three-blade rotor are different from each other and it can be concluded that the simulation should prevent using the single blade to represent the whole rotor especially when resonance of vibration is a crucial problem. The blade tip displacement in the Z direction experiences the largest amplitude because this direction faces the incoming flow and keeping large aerodynamic forces, ranging from 289Pa to 3.83MPa and a maximum deformation of 4.32mm under wind speed of 10m/s. Vibration Mode analysis results show that flap is the main type of vibration in first and second modes. The difference of natural frequency between single blade and blade rotor becomes larger as vibration mode increases.

3. The basic concept and the physical model of aeroelasticity have been applied on the aeroelastic stability investigation of 5kW wind turbine blade under different wind speeds. The expressions of different parameters such as aerodynamic stiffness and aerodynamic damping have been derived as well as the different aerodynamic force model. The lift and drag coefficient curves of S822 under wide range of angle of attack have been plotted with hybrid turbulence model. It can be observed from the aeroelastic stability analysis that the wind turbine will be stable when the wind speed is below 35m/s and will experience flutter when wind speed is over 35m/s. The phenomenon of frequency overlap has been observed in the analyzed result which is the main reason for wind turbine blade fluttering. The bending and torsion frequencies are 23.94Hz and 38.46 Hz respectively under TSR 3 and both move towards 30Hz under TSR 1.7. When TSR is smaller than 1.7 the wind turbine will experience flutter due to the overlapping of bending and torsion frequencies. The influence of mass center and different materials have been conducted and the results illustrate that moving the

mass center towards the twist center of airfoil as well as increasing the blade material stiffness will increase the critical flutter wind speed.

4. The disadvantage of the traditional designing method has been discussed and a better method which has thoroughly improved aerodynamic shape and power efficiency is being developed. Genetic algorithm has been selected to solve the multiple objectives optimization problem. The results show that chord according to optimized design is smaller than the initial design and the chord transition between different blade elements is more gentle and smoother than the initial design. The optimized twist angle is larger than the corresponding value of initial design. The comparison of the power coefficients and torque coefficients of wind turbine blade between the initial and optimized designs demonstrate that the multiple objectives optimization could improve wind turbine power and torque performances up to 4% and provide more electrical energy around 1073kW annually under rated wind speed. The aeroelastic analysis results illustrated that the optimized critical flutter wind speed of wind turbine blade will increase from 35m/s to 38m/s.

To sum up, the present study on wind turbine fluid structure interaction provides a comprehensive, reliable, accurate and fast numerical simulation method to predict and optimize wind turbine aerodynamic and structural performance under different working conditions. This method will help the wind turbine designers and engineers to quickly obtain more trustable wind turbine characteristics within a shorter period of time, with the less complicated and low cost computational resources and software. To adopt this new method the wind turbine industry could solve the wind turbine fluid structure interaction related problems more efficiently and economically.

7.2 Future works

Based on the present study and results obtained the following future works are proposed.

1. The wind profile in the present study is assumed as an uniform incoming wind velocity which could be replaced with the local atmospheric wind flow boundary layer profiles at different period of time throughout the year to improve the accuracy of the CFD prediction. The wind turbine power performance and aeroelastic stability obtained based on the local atmospheric wind flow boundary layer distributions will provide an optimal selection of wind turbine deployment in specific region.
2. The hollow wind turbine blade with spar structure could be used in future works to reduce the weight of wind turbine rotor. At the same time the coating film will be applied on wind turbine blade surface to keep blade clean and reduce wear and tear.
3. The effect of tower on the wind turbine should be taken into account in the future work. In the structure analysis and resonance investigation the effect of tower should be included in order to obtain a numerical simulation result closer to the real structure than before.
4. The other state-of-the-art hybrid turbulent approaches such as detached eddy simulation, delayed detached eddy simulation and zonal detached eddy simulation will be adopted and the simulation results will be compared in order to choose the best suitable hybrid turbulence model.
5. In future works the optimization method should take both suggestion in items (3) and (4) into account to provide practical and useful optimization information for wind turbine design in wind energy industry.

REFERENCES

Ansys helping documentation, Version 15, 2015.

Bak C, Fuglsang P, Sorensen NN, Madsen HA, Shen WZ and Sorensen JN, Airfoil Characteristics for Wind Turbines, Riso National Laboratory, 1999.

Bathe KJ, Zhang H and Ji S, "Finite element analysis of fluid flows fully coupled with structural interactions", Computers & Structures, Vol. 72, pp. 1-16, 1999.

Bazilevs Y, Hsu MC, Kiendl J, Wuchner R, and Bletzinger KU, "3D simulation of wind turbine rotors at full scale. Part II: fluid–structure interaction modeling with composite blades", International Journal for Numerical Methods in Fluids, Vol. 65, pp. 236-253, 2011.

Beards CE, Structural Vibration: Analysis and Damping, First ed.: Halsted Press, 1996.

Benra FK, "Numerical and experimental investigation on the flow induced oscillations of a single-blade pump impeller", Journal of Fluids Engineering, Vol. 128, pp. 783-793, 2006.

Bertin JJ and Smith ML, Aerodynamics for Engineers, First ed.: Prentice Hall, 1989.

Betz A, Introduction to the theory of flow machines, First ed.: Pergamon, 1966.

Bielawa RL, RotaryWing Structural Dynamics and Aeroelasticity Second ed.: American Institute of Aeronautics and Astronautics Inc, 2006.

Bisplinghoff RL, Ashley H and Halfman RL, Aeroelasticity, Dover ed.: Dover Publications Inc, 1996.

Bjorck A, "Dynamic stall and three-dimensional effects (DNS3D) Final Report of the JOU2-CT93-0345. Joule II Project", Report FFA TN 1995, Vol. 31, pp. 231-238, 1996.

Blazek J, Computational Fluid Dynamics: Principles and Applications, First ed.: Elsevier Science, 2001.

Boiko AV, Kirilovskiy SV, Maslov AA, and Poplavskaya TV, “Engineering modeling of the laminar–turbulent transition: Achievements and problems”, Journal of Applied Mechanics and Technical Physics, Vol. 56, pp. 761-776, 2015.

Bottega WJ, Engineering Vibrations, First ed.: CRC Press, 2006.

Brondsted P and Nijssen RPL, Advances in Wind Turbine Blade Design and Materials, First ed.: Woodhead Publishing Limited, 2013.

Burton T, Jenkins N, Sharpe D and Bossanyi E, Wind Energy Handbook, Second ed.: Wiley, 2011.

Cebeci T, and Smith AMO, Analysis of Turbulent Boundary Layers, First ed.: Academic Press, 1974.

Chaviaropoulos PK, “Flap lead-lag aeroelastic stability of W/T blades”, Wind Energy, Vol. 4, pp. 183-200, 2001.

Chaviaropoulos PK, Nikolaou IG, Aggelis KA, Soerensen NN, Johansen J, Hansen MOL, Gaunaa M, Hambræus T, VonGeyr HF, Hirsch C, Shun K, Voutsinas SG, Tzabiras G, Perivolaris Y and Dyrmoose SZ, "Viscous and aeroelastic effects on wind turbine blades. the VISCEL project. Part I: 3D Navier-Stokes rotor simulations", Wind Energy, Vol. 6, pp. 365-385, 2003.

Crighton DG, “The Kutta condition in unsteady flow”, Annual Review of Fluid Mechanics Vol. 17, pp. 411- 445, 1985.

Crimi p and Reeves BL, "Analysis of leading-edge separation bubbles on airfoils", AIAA Journal Vol. 14(11), pp. 1548-1555, 1976.

Davidson L and Dahlstrom S, "Hybrid LES-RANS: An approach to make LES applicable at high Reynolds number", International Journal of Computational Fluid Dynamics, Vol. 19(6), pp. 415-427, 2005.

Dewan A, Tackling Turbulent Flows in Engineering, First ed.: Springer, 2011.

Ding WJ, Self-Excited Vibration: Theory, Paradigms, and Research Methods, First ed.: Springer, 2011.

Donea J, Huerta A, Ponthot JP and Ferran AR "An arbitrary lagrangian-eulerian finite element method for transient dynamic fluid-structure interactions", Computer Methods in Applied Mechanics and Engineering, Vol. 33(1), pp. 689-723, 1982.

Dovgal AV, Kozlov VV and Michalke A, "Laminar boundary layer separation: instability and associated phenomena", Progress in Aerospace Sciences, Vol. 30, pp. 61-94, 1994.

Dowell EH, Peters DA, Clark R, Scanlan R, Cox D, Simiu E, Curtiss HCJ, Sisto F, Edwards JW, Hall KC and Strganac TW, A Modern Course in Aeroelasticity, Fourth ed.: Kluwer Academic Publishers, 2005.

Du Z, Selig MS, "The effect of rotation on the boundary layer of a wind turbine blade", Renewable Energy, Vol. 20, pp. 167-181, 2000.

Duquette MM and Visser KD, "Numerical implications of solidity and blade number on rotor performance of horizontal-axis wind turbines", Journal of Solar Energy Engineering, Vol. 125, pp. 425-432, 2003.

Ferziger JH and Peric M, Computational Methods for Fluid Dynamics, First ed.: Springer, 1999.

Franklin GF, Powell JD and Naeini AE, Feedback Control of Dynamic Systems, Sixth Ed.: Pearson, 2009.

Fung YC, An Introduction to The Theory of Aeroelasticity, Dover ed.: Dover Publications Inc, 1993.

Galdi GP and Rannacher R, fundamental trends in fluid-structure interaction Contemporary Challenges in Mathematical Fluid Dynamics and Its Applications - Vol. 1, First ed.: World Scientific Publishing, 2010.

Gasch R and Tvele T, Wind Power Plants - Fundamentals, Design, Construction and Operation, Second ed.: Springer, 2012.

Glauert H, "Airplane Propellers", Aerodynamic Theory, Vol. 4, pp. 169-360, 1935.

Goosen MF, Mahmoudi H and Ghaffour N, "Today's and future challenges in applications of renewable energy technologies for desalination", Critical Reviews in Environmental Science and Technology, Vol. 44(9), pp. 929-999, 2014.

GWEC, "Global Wind Report -Annual Market Update 2016", 2016.

Hand MM, Simms DA, Fingersh LJ, Jager DW, Cotrell JR, Schreck S, and Larwood SM, "Unsteady aerodynamics experiment phase VI: Wind tunnel test configurations and available data campaigns", Technical Report NREL/TP-500-29955, National Renewable Energy Laboratory, 2001.

Hansen A and Butterfield C, "Aerodynamics of horizontal-axis wind turbines", Annual Review of Fluid Mechanics, Vol. 25, pp. 115-149, 1993.

Hansen MH, “Aeroelastic instability problems for wind turbines”, *Wind Energy*, Vol. 10, pp. 551-577, 2007.

Hansen MOL, *Aerodynamics of Wind Turbines*, Second ed.: Earthscan, 2008.

Hau E, *Wind Turbines-Fundamentals, Technologies, Application, Economics*, Third translated ed.: Springer, 2013.

Houghton EL and Carpenter PW, *Aerodynamics for Engineering Students*, First ed.: Butterworth-Heinemann, 2003.

IEC, "Standard 61400-2 Wind Turbines-Part 2: Design Requirements for Small Wind Turbines ", Third ed.: International Electrotechnical Commission, 2013.

Ingram G, "Wind turbine blade analysis using the blade element momentum method", School of Engineering, Durham University, 2005.

Jamieson P, *Innovation in Wind Turbine Design*, First ed.: John Wiley & Sons Ltd, 2011.

Jureczko M, Pawlak M and Mezyk A, "Optimisation of wind turbine blades," *Journal of Materials Processing Technology*, Vol. 167, pp. 463-471, 2005.

Lawrence K, *Ansys Workbench Tutorial Release 14*, First ed.: SDC Publications, 2012.

Legay A, Chessa J and Belytschko T, “An Eulerian–Lagrangian method for fluid structure interaction based on level sets”, *Computational Methods in Applied Mathematics*, Vol. 195, pp. 2070-2087, 2004.

Leissa AW, Qatu MS, *Vibrations of Continuous Systems*, First ed.: McGraw-Hill, 2011.

Malcolm DJ, “Modal response of three-blade wind turbine”, *Journal of Solar Energy Engineering*, Vol. 124, pp. 372-377, 2002.

Manwell JF, McGowan JG and Rogers AL, Wind Energy Explained: Theory, Design and Application, First ed.: John Wiley & Sons Ltd, 2002.

Maskell EC, A Theory of the Blockage Effects on Bluff Bodies and Stalled Wings in a Closed Wind Tunnel, Reports and Memoranda No. 3400 of Aeronautical Research Council, 1963.

Meirovitch L, Fundamentals of Vibrations, Second ed.: McGraw-Hill New York, 2001.

Menter F, “Two-equation eddy-viscosity turbulence models for engineering applications”, AIAA Journal, Vol. 32(8), pp. 1598-1605, 1994.

Menter F and Egorov Y, “The scale-adaptive simulation method for unsteady turbulent flow predictions”, Flow, Turbulence and Combustion, Vol. 85, pp. 113-128, 2010.

Miller A, Muljadi E and Zinger DS, "A variable speed wind turbine power control", Energy Conversion, IEEE Transactions on, Vol. 12, pp. 181-186, 1997.

Min BY and Sankar LN, "Hybrid Navier-Stokes/free-wake method for modeling blade-vortex interactions", Journal of Aircraft, Vol. 47, pp. 975-983, 2010.

Pape AL and Lecanu J, “3D Navier–Stokes computations of a stall-regulated wind turbine”, Wind Energy, Vol. 7, pp. 309-324, 2004.

Petersen JT, Madsen HA, Bjorck A, Enevoldsen P, Oye S, Ganander H and Winkelaar D, “Prediction of dynamic loads and induced vibrations in stall”, Riso National Laboratory, 1998.

Pourrajabian A, Ashfar PAN, Ahmadizadeh M and Wood DH, “Aero-structural design and optimization of a small wind turbine blade”, Renewable Energy, Vol. 87, pp.837-848, 2016.

Pramod J, Wind Energy Engineering, First ed.: McGraw-Hill, 2011.

Qaisia AA and Bedoor BO, "Evaluation of different methods for the consideration of the effect of rotation on the stiffening of rotating beams", Journal of Sound and Vibration Vol. 280(3-5), pp. 531-553, 2005.

Rao SS, Vibration of Continuous Systems, First ed.: John Wiley & Sons Ltd, 2007.

Rasmussen F, Hansen MH, Thomsen K, Larsen TJ, Bertagnolio F, Johansen J, Madsen HA, Bak C and Hansen AM, "Present status of aeroelasticity of wind turbines", Wind Energy, Vol. 6, pp. 213-228, 2003.

Riziotis VA, Voutsinas SG, Politis ES, and Chaviaropoulos PK, "Aeroelastic stability of wind turbines: the problem, the methods and the issues", Wind Energy, Vol. 7, pp. 373-392, 2004.

Roache PJ, "Quantification of uncertainty in computational fluid dynamics", Annual Review of Fluid Mechanics, Vol. 29, pp. 123-160, 1997.

Rodi W, Constantinescu G and Stoesser T, Large Eddy Simulation in Hydraulics, First ed.: CRC Press, 2013.

Ronsten G, "Static pressure measurements on a rotating and a non-rotating 2.375 m wind Turbine blade comparison with 2D calculations", Wind Engineering and Industrial Aerodynamics, Vol. 39(1) , pp. 105-118, 1992.

Rugonyi S and Bathe KJ, "On finite element analysis of fluid flows fully coupled with structural interactions", Computer Modeling in Engineering and Sciences, Vol. 2, pp. 195-212, 2001.

Sagaut P, Deck S and Terracol M, Multiscale and Multiresolution Approaches in Turbulence LES, DES and Hybrid RANS/LES Methods: Applications and Guidelines, First ed.: Imperial College Press, 2013.

Sarrate J, Huerta A and Donea J, “Arbitrary Lagrangian-Eulerian formulation for fluid-rigid body interaction”, Computer Methods in Applied Mechanics and Engineering, Vol. 190 , pp. 3171-3188, 2001.

Selig MS and Granahan BD, "Wind tunnel aerodynamic tests of six airfoils for use on small wind turbines", Journal of Solar Energy Engineering, Vol. 126, pp. 986-1001, 2004.

Selig MS, Lyon CA, Giguere P, Ninham C and Guglielmo J, Summary of low-speed airfoil data, Vol. 1, SoarTech Publications, 1995.

Sessarego M, Dixon KR, Rival DE and Wood DH, “A hybrid multiobjective evolutionary algorithm for wind-turbine blade optimization”, Engineering Optimization, Vol. 47, pp.1043-1062, 2014.

Sessarego M and Wood DH, “Multi-dimensional optimization of small wind turbine blades”, Renewables: Wind, Water, and Solar, Vol. 2(9), pp.1-11, 2015.

Schlichting H, Boundary-Layer Theory, First ed.: McGraw-Hill, 1968.

Simms DA, Schreck S, Hand M and Fingersh L, “NREL unsteady aerodynamics experiment in the NASA-Ames wind tunnel: A comparison of predictions to measurements”, National Renewable Energy Laboratory, 2001.

Simon D, Evolutionary Optimization Algorithms, First ed.: Wiley, 2013.

Sivanandam SN and Deepa SN, Introduction to Genetic Algorithms, 2008th ed.: Springer, 2008.

Spera DA, Wind Turbine Technology, First ed.: CRC Press, 1994.

Takagi S, Sugiyama K, Ii S and Matsumoto Y, "A review of full eulerian methods for fluid structure interaction problems", Journal of Applied Mechanics, Vol. 79(1), pp. 1-7, 2012.

Takizawa K, and Tezduyar TE, "Multiscale space-time fluid-structure interaction techniques," Computational Mechanics, Vol. 48(3), pp. 247-267, 2011.

Tangler JL and Somers DM, "NREL airfoil families for HAWTs", National Renewable Energy Laboratory, 1996.

Tewari A, Aeroservoelasticity Modeling and Control, First ed.: Springer, 2015.

Tong W, Wind Power Generation and Wind Turbine Design, First ed.: Wit Press, 2010.

Verma G and Weber M, SolidWorks 2016 Black Book, Third Ed.: Create Space Independent Publishing Platform, 2015.

Versteeg HK and Malalasekera W, An introduction to Computational Fluid Dynamics: The Finite Volume Method, Second ed.: Prentice Hall, 2007.

Vries O, "On the theory of the horizontal-axis wind turbine", Annual Review of Fluid Mechanics, Vol. 15, pp. 77-96, 1983.

Wang XD, Fundamentals of Fluid–Solid Interactions: Analytical and Computational Approaches, First ed.: Elsevier, 2008.

Weekes S and Tomlin A, "Evaluation of a semi-empirical model for predicting the wind energy resource relevant to small-scale wind turbines", Renewable Energy, Vol. 50, pp. 280-288, 2013.

White FM, Viscous Fluid Flow, First ed.: McGraw-Hill, 1991.

Wilson RE and Lissaman PBS, "Applied aerodynamics of wind power machines", Academic Report, Oregon State University, Corvallis (USA) 1974.

Wilson RE, Lissaman PBS and Walker SN, "Aerodynamic performance of wind turbines", Oregon State Univ., Corvallis (USA) 1976.

Wilson SVR, Clausen PD and Wood DH, "Gyroscopic Moments on Small Wind Turbines Blades at High Yaw Rates", Australian Journal of Mechanical Engineering, Vol. 5(1), pp.1-9, 2008.

Wood DH, Small Wind Turbines: Analysis, Design, and Application, First ed.: Springer, 2011

Wright JR and Cooper JE, Introduction to Aircraft Aeroelasticity and Loads, First ed.: John Wiley & Sons Ltd, 2007.

Zhang PT and Huang SH, "Review of aeroelasticity for wind turbine: Current status, research focus and future perspectives", Front Energy, Vol. 5(4), pp. 419-434, 2011.

Zhao DP and Xu BQ, Wind Turbine Design Theory and Method, First ed.: Peking University Press, 2011.

Zimmerman RL and Olness FI, Mathematica for Physics, Second ed.: Addison-Wesley Longman Publishing Co, 2002.

APPENDIXES

Appendix A

S823 Airfoil

coordinate data (1)		coordinate data (2)		coordinate data (3)		coordinate data (4)	
x/c	y/c	x/c	y/c	x/c	y/c	x/c	y/c
1	0	0.25503	0.100219	0.049121	-0.076692	0.89846	0.004922
0.996182	0.001021	0.213826	0.094667	0.065425	-0.091707	0.934449	0.005687
0.985647	0.004487	0.17515	0.087661	0.084347	-0.102507	0.963294	0.004594
0.969834	0.009935	0.139404	0.079378	0.108058	-0.10978	0.9839	0.00254
0.949208	0.016186	0.106993	0.069983	0.136074	-0.114432	0.996032	0.000732
0.923311	0.02286	0.078238	0.059647	0.168058	-0.116517	1	0
0.892433	0.030245	0.053489	0.048562	0.203782	-0.116043		
0.857144	0.038305	0.032973	0.036935	0.243045	-0.113061		
0.818037	0.046896	0.017013	0.025059	0.285642	-0.107692		
0.775723	0.05581	0.005871	0.013252	0.331343	-0.100136		
0.73081	0.064793	0.001538	0.006021	0.379872	-0.090684		
0.68389	0.073566	0.000533	0.003322	0.430894	-0.079712		
0.635537	0.081835	0.000235	0.00216	0.483993	-0.067673		
0.586291	0.089312	0.000026	0.000734	0.538656	-0.055088		
0.536666	0.095725	0.000174	-0.001931	0.594258	-0.042513		
0.487145	0.100825	0.0008	-0.004837	0.650057	-0.03052		
0.438182	0.104394	0.002374	-0.009387	0.70519	-0.019649		
0.390202	0.106236	0.010193	-0.023971	0.758682	-0.010377		
0.343554	0.106115	0.021425	-0.041404	0.809471	-0.003071		
0.298349	0.104071	0.03433	-0.059382	0.856439	0.002045		

S823 airfoil shape and coordinate (Selig et al. 1995)

S822 Airfoil

coordinate data (1)		coordinate data (2)		coordinate data (3)		coordinate data (4)	
x/c	y/c	x/c	y/c	x/c	y/c	x/c	y/c
1	0	0.220471	0.082211	0.073374	-0.042563	0.93944	-0.001945
0.996089	0.000642	0.181021	0.076101	0.101374	-0.049059	0.96515	-0.000135
0.985048	0.003157	0.144635	0.069084	0.133441	-0.054581	0.984246	0.000448
0.968133	0.007602	0.111706	0.061261	0.169219	-0.059031	0.996022	0.000259
0.946046	0.013324	0.082528	0.052754	0.208482	-0.062361	1	0
0.918825	0.020084	0.057423	0.043713	0.250797	-0.064545		
0.887037	0.028045	0.036592	0.034302	0.295834	-0.065585		
0.851444	0.037037	0.02031	0.024706	0.343087	-0.065504		
0.812782	0.046731	0.008599	0.015142	0.392138	-0.06434		
0.771742	0.056677	0.00176	0.006074	0.44242	-0.062144		
0.728946	0.066315	0.000651	0.003396	0.493444	-0.058965		
0.684771	0.074819	0.000138	0.001358	0.544594	-0.054833		
0.638946	0.081988	0.000023	-0.000542	0.595407	-0.049596		
0.591971	0.087923	0.000294	-0.001935	0.645795	-0.043462		
0.544321	0.092477	0.000443	-0.002413	0.695149	-0.036874		
0.496467	0.095295	0.001282	-0.004385	0.742856	-0.030066		
0.44832	0.096342	0.005329	-0.010118	0.788279	-0.023032		
0.400324	0.095949	0.015376	-0.01868	0.831428	-0.016082		
0.353012	0.094259	0.030216	-0.027173	0.871626	-0.010003		
0.306932	0.091363	0.049559	-0.035201	0.907952	-0.005223		

S822 airfoil shape and coordinate (Selig and Granahan 2004)

Appendix B

Navier-Stokes equations for an unsteady, incompressible, three dimensional viscous flow:

$$X \text{ direction: } \rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = \rho f_x - \frac{\partial p}{\partial x} + \mu \nabla^2 u$$

$$Y \text{ direction: } \rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = \rho f_y - \frac{\partial p}{\partial y} + \mu \nabla^2 v$$

$$Z \text{ direction: } \rho \left(\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = \rho f_z - \frac{\partial p}{\partial z} + \mu \nabla^2 w$$

$$\text{Continuity equation: } \nabla \cdot \vec{V} = 0$$

Where u, v, w are velocities in x, y and z directions respectively, f_x, f_y, f_z , are body force in x, y and z directions respectively, ρ is density, μ is dynamic viscosity, p is pressure, t is time.

$$\vec{V} = u\vec{i} + v\vec{j} + w\vec{k},$$

$$\text{gradient operator } \nabla = \frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k}$$

$$\text{Laplace operator } \nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2}$$

Appendix C

Fast Fourier Transform

The definition of Fourier transform is as follow:

A signal $s(t)$ which is variation with time could be represented by a superposition of series of sinusoidal signal with different frequencies. The amplitude of the sinusoidal signal with frequency ν could be represented as below:

$$S(\nu) = \int_{-\infty}^{\infty} s(t) e^{-2\pi\nu t i} dt$$

Where i is the imaginary number, $S(\nu)$ is the frequency spectrum of signal $s(t)$.

The power spectrum density could be defined as:

$$P(\nu) = |S(\nu)|^2$$

which is used to measure the contribution of different frequency in the frequency spectrum.

The real part and imaginary part of $S(\nu)$ could be represented as follow:

$$Real = \int_{-\infty}^{\infty} s(t) \cos(2\pi\nu t) dt$$

$$Imaginary = \int_{-\infty}^{\infty} s(t) \sin(2\pi\nu t) dt$$

PUBLICATIONS

1. Zheng S, Chua LP and Zhao Y, “Simulations of Fluid-structure Interaction of a Wind Turbine”, Proceedings of the 3rd Symposium on Fluid-Structure-Sound Interactions and Control, pp. 407-413, 2015.
2. Zhao Y, Zheng S and Chua LP, “Chapter 11 Wind Turbines for Electricity Generation for Desalination: Design, Applications & Commercialization”, Renewable Energy Technologies for Water Desalination, First ed.: CRC Press, 2016.
3. Zheng S, Chua LP and Chai GB, “Using Whole Flow Field withHybrid RANS method to Investigate Fluid-Structure-Interaction of Wind Turbine”,Journal of Wind Engineering and Industrial Aerodynamics. (Ready to submit)
4. Zheng S, Chua LP and Chai GB,“Wind Turbine Aeroelastic Stability Evaluation and Optimization”, Renewable Energy. (Ready to submit)