

CRANFIELD UNIVERSITY

PAVANA KORAGAPPA

AERODYNAMIC DESIGN OPTIMIZATION OF LARGE-SCALE
OFFSHORE WIND TURBINE BLADE USING CFD

SCHOOL OF WATER, ENERGY AND ENVIRONMENT
PhD Energy and Power

PhD
Academic Year: 2021 - 2022

Supervisor: Dr Patrick Verdin
Associate Supervisor: Dr Ali Nabavi
May 2022

CRANFIELD UNIVERSITY

SCHOOL OF WATER, ENERGY AND ENVIRONMENT
PhD in Energy and Power

PhD

Academic Year 2021 - 2022

PAVANA KORAGAPPA

AERODYNAMIC DESIGN OPTIMIZATION OF LARGE-SCALE
OFFSHORE WIND TURBINE BLADE USING CFD

Supervisor: Dr Patrick Verdin
Associate Supervisor: Dr Ali Nabavi
May 2022

This thesis is submitted in partial fulfilment of the requirements for the
degree of PhD

© Cranfield University 2022. All rights reserved. No part of this
publication may be reproduced without the written permission of the
copyright owner.

Abstract

Renewable energy is expected to be the main source of power by 2050, bringing an end to the use of fossil fuels; this is the only way to achieve Net Zero. Wind turbines which majorly contribute to this agenda, not only help to reduce CO_2 emission, they are also environmentally friendly and form a cost-effective solution. The aerodynamic study and design of a wind turbine blade is essential as it is directly linked to the performance of the wind turbine. The maximum power generating wind turbine currently operating is the Haliade-X (GE) turbine, which has set a trademark at producing 14 MW, 13 MW or 12 MW. However, a need for higher power generating wind turbines is present to be able to reach the Net Zero target.

By upscaling the “DTU 10 MW Reference Wind Turbine” this research has achieved an aerodynamically stable 20 MW offshore wind turbine blade design. Variable rotation speed and variable pitch angle configurations have been considered to achieve an ideal power curve. The aerodynamic performance has been evaluated using CFD and quantified for a length optimized blade design. To ensure structural stability, chord and twist optimizations have also been performed.

The chord and twist of the designed blade have been optimized through the momentum theory and the blade element theory. 2D numerical simulations on FFA-W3 aerofoils used in the design of the wind turbine blade have been carried out initially to determine the angle of attack at minimum C_D/C_L ratios, which further helps to calculate the chord and twist of the blade. From the calculated value, a new design variant has been proposed and the aerodynamic performance has been evaluated using CFD.

Acknowledgement

I would want to take this opportunity to express my gratitude to everyone who has helped me tremendously throughout my PhD studies.

First and foremost, I shall be extremely thankful to my Supervisor Dr Patrick Verdin for recognising my expertise and providing me with the PhD opportunity and for his timely indispensable advice, knowledge, as well as numerous perceptive discussions and suggestions throughout my PhD which made this work possible. I'm also thankful for his unwavering commitment to our shared vision towards this research.

I would also like to show my sincere gratitude to my sponsors Social Welfare Department Karnataka, India for the financial support without which the dream of pursuing a PhD would not have been possible.

I wish to show my appreciation to my associate supervisor Dr Ali Nabavi. A special thanks to Dr Joao Amaral Teixeira for his scholarly advice during my initial stages of the PhD which gave a great boost to the research.

It is also my privilege to thank my loving husband Mr Sandeep Duvani Shivanagara for his constant moral support, encouragement during my research and for all the tasty food he prepared while I was busy focusing on my research.

I am also deeply grateful to my parents, siblings and in-laws for their trust, keen interest in my academic achievements. My diligent parents have given up their lives for my sisters and myself, and they have shown us unfailing love and care. I would not have gotten this far without their continuous support and prayers. When times got tough, I knew I could always count on my near and dear ones.

Last but not the least, a sincere thanks to Cranfield University for providing an excellent platform and exclusive resources which enabled me to pursue my PhD with excellence.

Table of Contents

Abstract	iii
Acknowledgement	iv
Table of Contents	v
List of Figures	ix
List of Tables	xiii
Nomenclature	xv
List of Abbreviations	xvi
1 Introduction	1
1.1 Roadmap	5
1.2 Aim and Objectives	7
1.3 Methods and Methodology	8
2 Literature Review	10
2.1 Background Theory	10
2.2 Principles of Wind Energy Conversion	12
2.2.1 The Betz Elementary Momentum Theory.....	12
2.2.2 Power Calculation Using Aerodynamic Lift or Drag	16
2.2.3 Comparison of Rotors Using Aerodynamic Lift and Drag	19
2.2.4 Aerofoil Theory	21
2.2.5 Velocity Triangles	22
2.2.6 Wind	24
2.3 Rotor Aerodynamics	25
2.3.1 Blade Element Momentum Theory	25
2.3.2 Rotor Power Characteristics	26
2.3.3 Power and Torque Characteristics	27
2.3.4 Aerodynamic Power Control	28
2.3.5 Power Output Prediction	29
2.3.6 Power Control by Rotor Blade Pitching	30
2.3.7 Aerodynamic Characteristics of an Aerofoil	31
2.3.8 Boundary Layer	31
2.4 Blade Design for HAWT	32

2.4.1	Tip Speed Ratio (TSR)	32
2.4.2	Shape and Quantity of Blade Plans	34
2.4.3	Configuration	35
2.4.4	Aerodynamics	36
2.4.5	Angle of Twist	37
2.4.6	Blade Thickness	38
2.4.7	Off-Design Conditions and Power Regulation	38
2.4.8	Smart Blade Design	40
2.4.9	Control Methods	41
2.4.10	Control Strategies	43
2.6	Critical Review of Literature	45
2.7	Summary of literature review	55
3	Methodology	57
3.1	Introduction	57
3.2	Turbulent Flow Modelling for Wind Turbines	58
3.2.1	Reynolds-averaged Navier-Stokes equations for CFD	58
3.2.2	The Spalart-Allmaras model	59
3.2.3	The k- ϵ model	59
3.2.4	The k- ω SST model	60
3.3	Wall Function for a Turbulent Model	61
3.4	Mesh Generation Strategy	62
3.5	Two-Dimensional CFD	63
3.5.1	Introduction	63
3.5.2	Aerofoil Geometry	63
3.5.3	Flow Domain Modelling	64
3.5.4	Grid Generation using ICEM-CFD	65
3.5.5	Boundary Conditions and Solver Settings	66
3.5.6	Grid Sensitivity Study	67
3.5.7	Comparison and Discussion	69
3.6	Three-Dimensional CFD	73
3.6.1	DTU 10 MW RWT Design Specification	74
3.6.2	Wind Turbine Blade Geometry	74

3.6.3	Flow Domain Modelling and Sensitivity Study	76
3.6.4	Grid Generation	79
3.6.5	Solver Setting and Boundary Condition	82
3.6.6	Grid Independence Study	86
3.6.7	Baseline Wind Turbine Blade Performance	89
3.6.8	Comparison and Discussion	95
4	Results and Discussion	98
4.1	Introduction	98
4.2	Upscaling of Wind Turbines	98
4.3	Upscaling Methodology for Length Optimization	100
4.4	Upscaling the Blade Length Using QBlade	100
4.4.1	Upscaled Blade Geometry	101
4.4.2	Results	103
4.5	Upscaled Blade Length Study Using CFD	107
4.5.1	Meshing	107
4.5.2	Results	108
4.5.3	Discussion	116
4.6	Upscaling Methodology for Chord Length and Twist Angle Optimization	119
4.7	Upscaling Blade Chord Length and Twist Angle Using QBlade	122
4.7.1	Upscaled Blade Geometry	124
4.7.2	Results	126
4.8	Upscaled Blade Chord Length and Twist Angle Study Using CFD	129
4.8.1	Meshing	130
4.8.2	Results	131
4.8.3	Discussion	139
4.9	Comparison	142
5	Conclusions and Future work	145
5.1	Conclusions	145
5.2	Recommendations for Future Work	148
6	References	150
7	Appendix	164

7.1	Boundary Condition Interaction and Placement Sensitivity for Baseline Geometry	164
7.2	Pitch Sensitivity Study for Baseline Geometry	165
7.3	Pitch Sensitivity Study for Length Optimization	166
7.4	Pitch Sensitivity Study for Chord Length and Twist Angle Optimization	167

List of Figures

Figure 1.1	PhD research roadmap	6
Figure 1.2	Study framework of large-scale offshore wind turbines	7
Figure 2.1.1	Shaft and rotor orientation configurations [18]	11
Figure 2.2.1	Free stream air flow according to Betz wind turbine [18]	13
Figure 2.2.2	C_p versus $\frac{v_2}{v_1}$ where v_2 is the downstream velocity and v_1 is the upstream velocity [13]	15
Figure 2.2.3	Ideal disk-shaped energy converter depicting flow conditions while maximum power is being extracted [13]	16
Figure 2.2.4	Aerodynamic forces on an aerofoil [13]	17
Figure 2.2.5	Representation of drag device experiencing aerodynamic forces and flow condition [13]	18
Figure 2.2.6	Comparison between (a) drag and (b) lift driven rotor [23]	19
Figure 2.2.7	Diagram illustrating lift, drag, and angle of attack on NACA 0012	21
Figure 2.2.8	Velocity triangle of wind turbine aerofoil cut section [7]	23
Figure 2.2.9	Atmospheric boundary layer [23]	24
Figure 2.3.1	Aerofoil cross section of a blade element experiencing dynamic forces and velocities [13]	26
Figure 2.3.2	Power characteristics of experimental WKA-60 wind turbine rotor [13]	28
Figure 2.3.3	Wind turbine power curve [34]	29
Figure 2.3.4	Feather or stall pitching to control the rotor input power [13] .	30
Figure 2.3.5	Laminar and turbulent boundary layers [22]	32
Figure 2.4.1	Blade form representation with regional classification [18]	35
Figure 2.4.2	Modern large-scale wind turbine configuration [18]	36
Figure 2.4.3	A current HAWT blade featuring an increased linear chord length, various aerofoil shapes, and twist angle [18]	41
Figure 2.4.4	Pitch (left), yaw (right) adjustment [52]	42
Figure 2.4.5	Power curves for various control strategies (VS-VP, being	

	the ideal curve) [52]	44
Figure 2.6.1	Mechanical Power versus Wind speed curves for (i) Tells and (ii) Nordtank [56]	46
Figure 2.6.2	Lift coefficient for an aerofoil with 18% thickness, at $R_e = 9 * 10^6$ [62]	48
Figure 2.6.3	Thrust versus wind speed result comparison [70]	51
Figure 2.6.4	Calculated power versus wind speed comparison for different winglet design [77]	52
Figure 3.3.1	Regional classification of near wall turbulence flow [119]	61
Figure 3.5.1	FFA-W3 series aerofoils used for the 2D studies	64
Figure 3.5.2	2D fluid domain	65
Figure 3.5.3	Mesh generated in the domain and around the aerofoil	65
Figure 3.5.4	Comparison of lift coefficient and moment coefficient variation with grid size of FFA-W3-241 aerofoil	68
Figure 3.5.5	Pressure contour for grid size (a) 43,360 (b) 80,540 (c) 134,184 and (d) 247,844	68
Figure 3.5.6	Pressure coefficient for different grid size	69
Figure 3.5.7	Lift coefficient vs angle of attack plots for various FFA-W3 series	72
Figure 3.6.1	Baseline wind turbine blade geometry	75
Figure 3.6.2	Blade geometry for CFD simulation	76
Figure 3.6.3	Conical and cylindrical shaped domain	77
Figure 3.6.4	Dimensions of CFD domain - (a) Side view - (b) front view, D-diameter, and R-radius of the blade	78
Figure 3.6.5	Mesh on (a) subdomain and (b) outer cylindrical flow domain	81
Figure 3.6.6	Blade mesh and mesh around blade aerofoil at an intermediate section (80% span)	82
Figure 3.6.7	Convergence plots	86
Figure 3.6.8	Thrust and power coefficient results comparison with grid size at a wind speed 11 m/s	87

Figure 3.6.9	Comparison of pressure contours for (a) 30% span and (b) 60% span of the blade for grids of (I) 2.57 million and (II) 4.64 million cells	88
Figure 3.6.10	Comparison of pressure coefficient for (a) 30% span and (b) 60% span of the blade for grids of (I) 2.57 million and (II) 4.64 million cells	89
Figure 3.6.11	C_p vs TSR at wind speed of 11 m/s (QBlade)	90
Figure 3.6.12	Thrust and power variation with different pitch angle at velocity 20 m/s	93
Figure 3.6.13	Thrust and power variation with different pitch angle at velocity 25 m/s	95
Figure 3.6.14	Thrust coefficient and power coefficient results comparison ..	97
Figure 4.4.1	Blade geometry for upscaled blade	102
Figure 4.4.2	Power coefficient vs tip speed ratio for a upscaled blade	103
Figure 4.4.3	Power and thrust versus velocity plot	105
Figure 4.4.4	Power coefficient and thrust coefficient versus velocity plot ..	106
Figure 4.5.1	Upscaled blade geometry	107
Figure 4.5.2	Mesh on blade geometry	108
Figure 4.5.3	Pressure contour plots for pitch angle (I) 8.9° and (II) 8.65° at (a) 30% and (b) 60% of the span	110
Figure 4.5.4	Pressure coefficient plots for pitch angle 8.9 and 8.65 at (a) 30% and (b) 60% of the span	111
Figure 4.5.5	Pressure coefficient C_p and pressure contour plots for 16.83 velocity 25 m/s for different pitch angle (a) 16.93 degree (b) degree (c) 16.68 degree at span 30%	113
Figure 4.5.6	Pressure coefficient C_p and pressure contour plots for 16.83 velocity 25 m/s for different pitch angle (a) 16.93 degree (b) degree (c) 16.68 degree at span 60%	114
Figure 4.5.7	Aerodynamic results from CFD study	116
Figure 4.5.8	Aerodynamic results comparison between baseline ("DTU 10 MW RWT"), QBlade, and CFD	119
Figure 4.6.1	Aerodynamic forces, velocities, and angles at blade tip [123]	120

Figure 4.7.1	Comparison of chord length and shape of the blade	124
Figure 4.7.2	Blade geometry for chord and twist optimization	125
Figure 4.7.3	Power coefficient vs tip speed ratio for chord and twist optimization	125
Figure 4.7.4	Power and thrust versus velocity comparison	128
Figure 4.7.5	Power coefficient and thrust coefficient versus velocity comparison	129
Figure 4.8.1	Final blade geometry	129
Figure 4.8.2	Mesh on the blade, at a section, and at a cross section	130
Figure 4.8.3	Pressure coefficient C_p and pressure contour plots for velocity 20 m/s for different pitch angle (a) 15.72 <i>degree</i> (b) 15.62 <i>degree</i> (c) 15.57 <i>degree</i> (d) 15.47 <i>degree</i> at span 30%	133
Figure 4.8.4	Pressure coefficient C_p and pressure contour plots for velocity 20 m/s for different pitch angle (a) 15.72 <i>degree</i> (b) 15.62 <i>degree</i> (c) 15.57 <i>degree</i> (d) 15.47 <i>degree</i> at span 60%	134
Figure 4.8.5	Pressure coefficient C_p and pressure contour plots for velocity 25 m/s for different pitch angle (a) 20.57 <i>degree</i> (b) 20.42 <i>degree</i> (c) 20.27 <i>degree</i> at span 30%	136
Figure 4.8.6	Pressure coefficient C_p and pressure contour plots for velocity 25 m/s for different pitch angle (a) 20.57 <i>degree</i> (b) 20.42 <i>degree</i> (c) 20.27 <i>degree</i> at span 60%	137
Figure 4.8.7	Chord and twist optimization results	139
Figure 4.8.8	Aerodynamic results comparison of chord length and twist angle optimization	142
Figure 4.9.1	Thrust and power versus velocity curve for design variants ..	143
Figure 4.9.2	Power curve comparison of different wind turbine models	144

List of Tables

Table 2.4.1	Tip speed ratio design implications[18]	33
Table 3.5.1	FLUENT solver settings	66
Table 3.5.2	Reynolds numbers used for respective aerofoils	69
Table 3.6.1	Baseline wind turbine blade design specification[12]	74
Table 3.6.2	Baseline wind turbine blade geometry details[12]	75
Table 3.6.3	Boundary condition interaction and placement sensitivity	79
Table 3.6.4	Solver parameter settings used in FLUENT	85
Table 3.6.5	Operational parameters for the baseline blade geometry, (pitch-nose down) [12]	91
Table 3.6.6	Optimized pitch in current study (CFD)	91
Table 3.6.7	Percentage of error for thrust and power at velocity 20 <i>m/s</i> ...	93
Table 3.6.8	Percentage of error for thrust and power at velocity 25 <i>m/s</i> ...	95
Table 3.6.9	Percentage of error between three software	96
Table 4.2.1	Properties for 5 to 20 MW wind turbine using simple scaling approach [122]	99
Table 4.4.1	Basic key parameters of 10 and 20 MW WT	101
Table 4.4.2	Upscaled wind turbine blade geometry parameters	102
Table 4.4.3	Operational parameters and aerodynamic results at different velocities (the * symbols indicate the velocities to be simulated in CFD)	104
Table 4.5.1	Pitch angle optimization for higher velocities	109
Table 4.5.2	Pitch angle sensitivity study for velocity 16 <i>m/s</i>	110
Table 4.5.3	Pitch angle sensitivity study for velocity 25 <i>m/s</i>	112
Table 4.5.4	Aerodynamic results from CFD for length optimization	115
Table 4.5.5	Percentage of error between QBlade and CFD results	117
Table 4.7.1	Calculated chord and twist of the current design	123
Table 4.7.2	Operational parameter and aerodynamic results at different velocities	127

Table 4.8.1	Pitch angle sensitivity study	131
Table 4.8.2	Pitch angle optimization carried out for 20 <i>m/s</i>	132
Table 4.8.3	Pitch angle optimization carried out for 25 <i>m/s</i>	135
Table 4.8.4	Aerodynamic results from CFD for chord length and twist angle optimization	138
Table 4.8.5	Percentage of error between QBlade and CFD results	140
Table 7.1	Different turbulence models and time setting cases at velocity 25 <i>m/s</i>	164
Table 7.2	Percentage of error for thrust and power at velocity 5 <i>m/s</i>	165
Table 7.3	Percentage of error for thrust and power at velocity 12 <i>m/s</i> ...	165
Table 7.4	Percentage of error for thrust and power at velocity 16 <i>m/s</i> ...	166
Table 7.5	Percentage of error for thrust and power at velocity 12 <i>m/s</i> ...	166
Table 7.6	Percentage of error for thrust and power at velocity 20 <i>m/s</i> ...	167
Table 7.7	Percentage of error for thrust and power at velocity 5 <i>m/s</i>	167
Table 7.8	Percentage of error for thrust and power at velocity 12 <i>m/s</i> ...	168
Table 7.9	Percentage of error for thrust and power at velocity 16 <i>m/s</i> ...	168

Nomenclature

a	Axial induction factor	R_e	Reynolds number
A	Cross-sectional area (m^2)	$R_{e\,crit}$	Critical Reynolds number
a'	Tangential induction factor	s	Swept area (m^2)
B	Blade quantity	sf	Scaling factor
c	Speed of sound (m/s)	v'	Flow velocity (m/s)
C	Chord length (m)	v_1	Free stream velocity (m/s)
C_D	Drag coefficient	v_2	Angular velocity (m/s)
C_L	Lift coefficient	$\dot{V}$	Volume flow (m^3/s)
$C_{L\,max}$	Maximum lift coefficient	v_{rel}	Relative velocity (m/s)
C_{MR}	Torque coefficient	v_{wd}	Design wind speed (m/s)
C_{opt}	Optimum chord length (m)		
C_P	Power coefficient	Greek symbols	
C_{Pmax}	Maximum power coefficient	ρ	Density (kg/m^3)
C_t	Thrust coefficient	λ	Tip speed ratio (TSR)
C_{PR}	Rotor power coefficient	α	Angle of attack (AOA)
D	Drag (N)	μ	Viscosity ($kg/m.s$)
E	Kinetic energy (Nm)	Ω	Angular speed (m/s)
L	Lift (N)	φ	Angle of relative velocity (<i>degree</i>)
l	Length (m)	θ_p	Pitch angle (<i>degree</i>)
$\dot{m}$	Mass flow rate (kg/s)	$\theta_{p,0}$	Pitch angle at the tip (<i>degree</i>)
m	Mass (kg)	ε	Yaw error angle (<i>degree</i>)
M	Torque (Nm)	θ_T	Twist angle (<i>degree</i>)
ΔP	Power losses		
P	Power (W)		
P_0	Power of the wind (W)		
P_R	Rotor power (W)		
r	Radius at a distance along the blade span (m)		

R Blade radius (m)

Abbreviations

AEP	Annual Energy Production
AOA	Angle of Attack
BEM	Blade Element Momentum
BC	Boundary Condition
BMT	Betz' Momentum Theory
CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
DES	Detached Eddy Simulation
DNS	Direct Numerical Simulation
DTU	Technical University of Denmark
DU	Delft University
DOWEC	Dutch Offshore Wind Energy Converter Project
FAST	Fatigue, Aerodynamics, Structures, and Turbulence
FFA	Flygtekniska Forsöksanstalten Airfoils
FS-FP	Fixed Speed - Fixed Pitch
FS-VP	Fixed Speed - Variable Pitch
GE	General Electric
HAWT	Horizontal Axis Wind Turbines
HAWTOpt2	Horizontal Axis Wind Turbine Optimization Tool
HRLM	Hybrid RANS-LES Methods
IEA	International Energy Agency
KE	Kinetic Energy
LES	Large Eddy Simulation
NACA	National Advisory Committee for Aeronautics
NREL	National Renewable Energy Laboratory

RANS	Reynolds Average Navier-Stokes
RECOFF	Recommendations For Design Of Offshore Wind Turbines Project
RNG	Renormalization Group
RSM	Reynolds Stress Model
RWT	Reference Wind Turbine
SA	Spalart-Allmaras
SST	Shear Stress Transport
TMASO	Torque Matched Aerodynamic Shape Optimization
TSR	Tip Speed Ratio
VAWT	Vertical Axis Wind Turbines
VS-FP	Variable Speed – Fixed Pitch
VS-VP	Variable Speed - Variable Pitch
WindPACT	Wind Partnerships For Advanced Component Technology Project
WMB	Wind Multi Block
WT	Wind Turbine

1. Introduction

Since medieval times around 5000 years ago, wind has been used as the source of energy for carrying out various actions. There is a history of wind turbines being used for agriculture, and household purposes such as milling. Sailboats also used wind energy to navigate through the seas. Dating back to 200 B.C. people from China and Persia used wooden wind powered water mills for pumping and grain grinding. Wind energy has also been used as the source of energy for irrigation, water pumping in the olden days. However, in the recent era, as the electricity production developed, electric lines started encroaching every field, confining wind powered works to rural areas and causing the first phase of decline for wind use. Later, during the 19th century there was a shift from coal to petroleum.

Numerous scientists and researchers considered the damaging impacts of fossil fuels, petroleum, and nuclear, and saw the wind energy and natural gases as the potential future power generating. Wind power, as an alternative to fossil fuels, petroleum and nuclear power is available in abundance, it is also renewable and environmentally friendly.

Wind is described as the flow of gases in large scale, which is often referred by the strength and direction of the flow. According to the metrology science, short bursts of air are termed 'gust', and strong winds which have an intermediate duration are termed 'squalls.' The long duration high strength wind is called by various terminologies depending on the strength and duration as 'gale', 'storm' and 'hurricane.' The main cause for atmospheric wind generation is atmospheric pressure difference due to which air moves from the higher-pressure region to the lower pressure region, ultimately resulting in the formation of wind streams of various speeds. Also, the temperature gradients form a factor causing the generation of wind. Land and sea have their own significant temperature characteristics. Since wind is also generated due to changes in temperature generally, more wind is available in the sea regions.

A wind turbine is a mechanical device which uses wind as the feed and converts the kinetic energy in it to electrical energy using various intermediate devices like gearbox (low-speed, High-speed variants), and shaft. Rotor blade, nacelle, tower, and generator are commonly found components in a wind turbine.

For the last decades wind energy has been very important due to the green energy revolution, and research and development have been carried out. Offshore wind turbines show an increasing impact towards reaching Net-Zero by 2050. It is now crucial to consider improving wind turbines further and improve their design and efficiency at lower cost [1].

There are many ways to improve the wind energy. A simple way would be to improve the efficiency of existing wind turbines by attaching a winglet at the tip of their blade, however, the improvement would be minimal [2]. A second option would be to extend any wind farm by placing additional wind turbines of same efficiency to generate additional power. A third option would be to design a large-scale wind turbine which would produce a large amount of power. There are many aspects while designing a large-scale wind turbine, mainly related to the structural and aerodynamic performance. An efficient structural design would reduce the material and maintenance costs of the turbine, and would improve its aerodynamic efficiency, generating a higher amount of power [3].

The current research aims to produce a 20 *MW* horizontal axis wind turbine blade design, which is the highest power reached to date worldwide. Increasing the power beyond 20 *MW* would increase the size of the overall turbine which would be unfeasible due to issues like weight and corresponding loads. Large scale offshore wind turbines have a significant impact on the growing economy, the wind being a source of renewable energy. The use of such resources does not highly affect the environment, they are also known to be eco-friendly. Designing a large-scale wind turbine which can produce 20 *MW* power, will help for the daily commercial utilization of power. The conversion of kinetic energy into mechanical energy has been studied and improved for decades and maximizing the utilization of the kinetic energy of the wind is/was one of the objectives of this work.

Due to its wide availability, commercially available technology, low cost, and significant contribution to CO_2 reduction, wind energy plays a critical role in the development of sustainable energy. In both onshore and offshore wind projects, wind turbines account for most of the costs. As a result, wind turbine aerodynamics is the backbone of the wind energy industry because its performance is directly tied to it [4].

Blade Element Momentum theory (BEMT) has been traditionally used to develop the rotor blades of wind turbines and assess their performance behaviour [5]. Many studies have been carried out on small scale horizontal axis wind turbines using BEMT, Computational Fluid Dynamics (CFD) and experimental methods [5][6][7]. Fewer studies have been performed on commercial large scale offshore wind turbines, where further research and development is required, due to their capacity to produce a high amount of electricity/power.

A wide range of studies have been performed on rotor blades aerofoils [8][9]. The higher the thickness the better the structural properties, but this also affects blades' aerodynamic efficiency. Hence the blade thickness is limited to 21%-27%. Flygtekniska Forsöksanstalten Airfoils (FFA) developed by the Aeronautical Research Institute of Sweden and Delft University (DU) are the preferred aerofoils because of the thickness and lift to drag ratio they can achieve [10].

The research on vortex reduction (or wake effect) and turbulence level showed that designing and optimizing the winglet does not contribute much to the power coefficient. Varying the pitch and flap design for a large wind turbine makes the design complex and expensive, and manufacturers require a simple design for ease of fabrication.

A “5 MW Reference Wind Turbine” was designed to overcome all these issues in 2009 [11]. Later, in 2011 [3] and 2013 [12], a “10 MW Reference Wind Turbine” was designed. The classical similarity rules were used to up-scale the rotor blades.

One key parameter used to up-scale a wind turbine blade is the blade diameter. However, increasing the diameter has an adverse effect on the mass. A large wind turbine means a large nacelle, and thus the added weight makes the size of the tower to increase. Hence, it is essential to design large, cost-effective, and reliable offshore wind turbines, which produce a large amount of power. This can be achieved by varying parameters such as the length, chord, thickness, twist, RPM, and pitch of the wind turbine blades, the increase of aerodynamic performance of the blades, producing highly efficient wind turbines. To achieve the expected results, a domain/geometry and pitch sensitivity study must be carried out.

In the present research, a better performing large scale offshore wind turbine has been designed through varying critical aerodynamic parameters using CFD. CFD offers flexibility in terms of design changes, since it can be modified as per the needs several times finally arriving at an optimal design, it is cost effective, and it further allows the user to visualize forces acting on even the smallest part of the design which is advantageous. Simulations have been carried out for different cases. The main aim of the research is to obtain efficient eco-friendly renewable energy through a maximum utilization of the Kinetic Energy present in the free-stream wind.

To achieve the desired aim of the research, the thesis is divided into six chapters, with a brief overview of each of them provided below.

Chapter 1 - Introduction: Subject and importance of the research are explained briefly along with the research gap. Current problems faced due to fossil-fuel and solutions are described. Contribution to knowledge through this research work is presented and discussed.

The aim and objectives of the PhD research project are also provided in this chapter, and the methods used to achieve the aim are explained.

Chapter 2 - Literature Review: A literature survey is carried out on large scale offshore wind turbines and the associated design parameters. The survey helps to understand current and future requirements for operators to install and design offshore wind turbines and be able to efficiently harvest energy from the

wind. Current wind turbine blade designs are discussed, numerical and experimental methods used for the design of offshore wind turbines are also critically discussed. This chapter also includes principles of wind energy, rotor aerodynamics, horizontal axis wind turbine blade design.

Chapter 3 - Methodology: A baseline model and its performance parameter are explained. In this section 2D and 3D CFD methodologies that were employed in this study are discussed. The purpose of this research is to find the best CFD approach and methodologies for a large-scale offshore wind turbine. Grid independence study, boundary condition (BC) interactions and BC placement sensitivity analysis are explained along with BC type, solver settings and domain sizes. A pitch sensitivity study is also explained. Results obtained through the current research (CFD), QBlade and EllipSys3D (tool used by DTU) are compared.

Chapter 4 - Results / Findings: A novel design variants methodology is explained. Simulation results of the baseline design and new design variants for an offshore wind turbine blade are compared and discussed. Flow physics is shown at different sections of the blade.

Chapter 5 - Conclusions and Future work: The outcomes of the PhD work are summarised in this chapter. This chapter also provides suggestions for further research and development in the field.

1.1 Roadmap

Figure 1.1 shows the roadmap followed to successfully achieve the desired aim and objectives of the research. Through textbooks, theory, and principles, wind turbines were studied. The current problems faced by industries to improve the power efficiency were discussed. Through a literature review, by referring to journal papers, conference proceedings and current up to date development news, the research gap was defined. The “DTU 10 MW Reference Wind Turbine” was selected as the baseline model since its geometry is freely available from the literature. The required geometrical parameters and

aerodynamic results were also collected from the literature. The baseline model is an upwind rotor blade with a radius of 89.166 *m*.

A 2D CFD study was carried out to understand the flow around the aerofoils and its behaviour. A 3D baseline model was designed, and a domain and pitch sensitivity studies were carried out. The current CFD results were compared with existing results from the literature and with QBlade results. Along with these two above-mentioned studies, a grid independence study was carried out for both 2D and 3D models.

The aerodynamic parameters of the baseline model were then changed, based on Betz' formulation, and new design variants were obtained. The design variants were then modelled, and the mesh was generated. A detailed study was carried out to finalise the results. Finally, a 20 MW offshore wind turbine blade was generated.

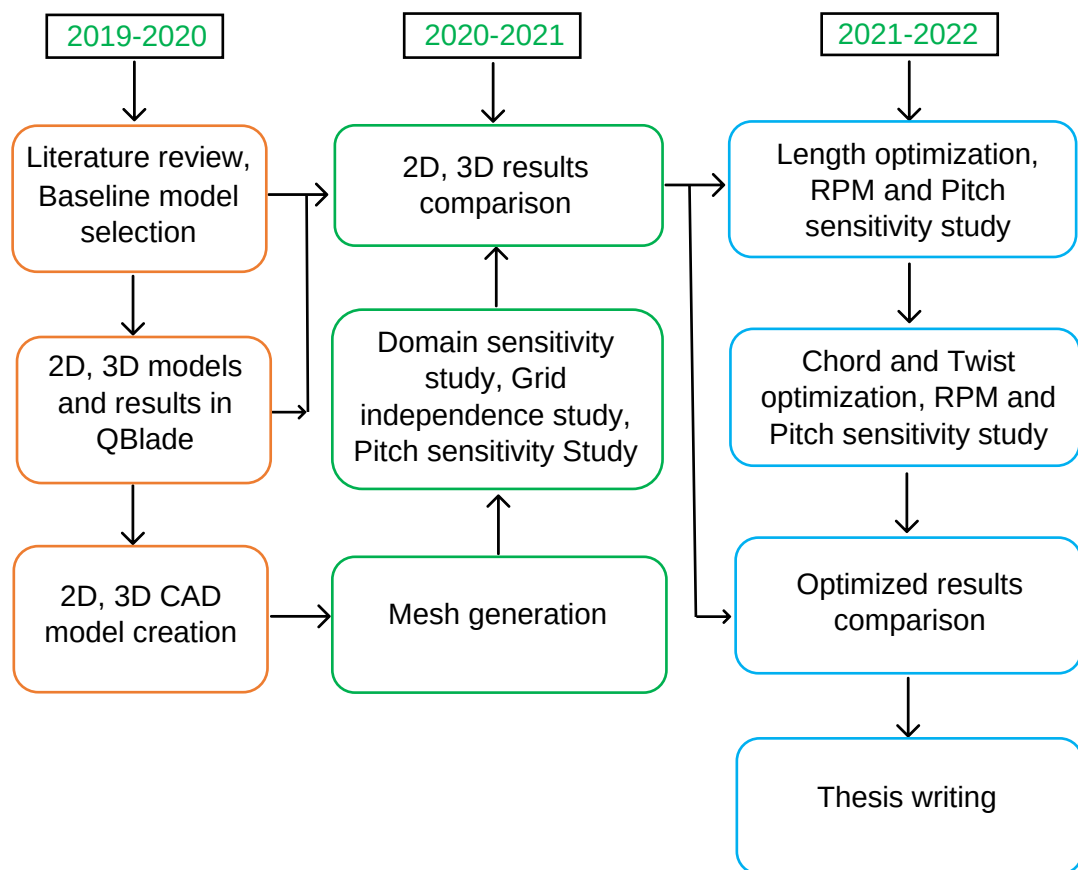


Figure 1.1: PhD research roadmap

1.2 Aim and Objectives

Large scale offshore wind turbines have mainly three key designs: 1) Aerodynamic design, 2) Structural design, 3) Aero-Servo-Elastic Design. Figure 1.2 shows the framework for the study of large-scale offshore wind turbines. The aerodynamic design consists of aerofoil selection, rotor design, and aerodynamic performance. The structural design is dedicated to analysing the material, load acting, buckling and strength of the blade and tower. Finally, the task of the aero-servo-elastic design is to describe the characteristic of the components like nacelle, hub, and tower and drivetrain properties and its aero-elastic stability.

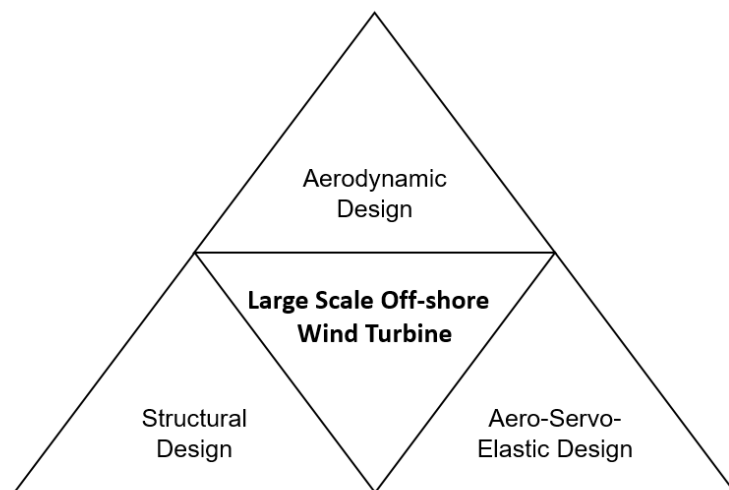


Figure 1.2: Study framework of large-scale offshore wind turbines

The present research involves the study of the aerodynamic design. The aim is to design and analyse an offshore wind turbine and improve its aerodynamic performance under 'design' and 'off-design' conditions, using CFD. This research will build a foundation to carry out further research on structural and aero-servo-elastic design. The aerodynamic design involves two main sections 1) 2D aerofoil study, and 2) 3D blade design. Initially, according to the design requirements, a set of aerofoils will be selected, and their characteristics will be analysed to build the 3D blade geometry.

To have a better performing rotor blade, the following objectives are defined:

1. To assimilate the principles and techniques that influence the aerodynamic performance of a wind turbine through a literature review.
2. To select a suitable baseline offshore wind turbine and its operating conditions.
3. To obtain the aerodynamic performance of a baseline offshore wind turbine.
4. To investigate the effects of different wind turbine parameters on the aerodynamic performance.
5. To achieve a 20 *MW* large scale offshore wind turbine.

1.3 Methods and Methodology

The methods and methodology of the research are listed below:

- A literature review on aerodynamic design and performance of offshore wind turbines was carried out by referring to Journal papers, conference proceedings, and current up to date development news.
- The “DTU 10 MW reference wind turbine” has been selected as the baseline model. The data files have been collected from the DTU designers.
- The geometrical model of the “DTU 10 MW reference wind turbine” was built using SOLIDWORKS and a CFD analysis was carried out using ANSYS FLUENT after performing a grid independence study.
- The design and off-design performances of the “DTU 10 MW reference wind turbine” was carried out and the results of EllipSys3D (tool used by DTU), were compared with the results of the CFD simulation.
- The blade was redesigned for the 20 *MW* power by varying the length of the blade. A similarity rule was applied to achieve the required blade length.

- The optimum RPM was achieved for each individual velocity between 5 *m/s* and 25 *m/s* by running a set of simulations.
- The chord to thickness ratio and the twist angle of the aerofoil were modified with respect to the new length to generate a new design variant using Betz' optimum rotor design.
- Operational parameters in particularly pitch must be varied to achieve the optimised results.
- Geometric modelling and CFD simulations were carried out for each of the design variants using SOLIWORKS and ANSYS FLUENT, respectively.
- The performances of different wind turbine design variants were compared in terms of power coefficient and thrust coefficient against wind speed.
- The design of a 20 *MW* large scale offshore wind turbine was based on improved aerodynamic performance and operational parameters.

2. Literature Review

2.1 Background Theory

For hundreds of years, wind has been utilised to generate power, but historical designs (windmills) were inefficient, massive, and heavy. Windmills were replaced by fossil fuel engines in the early nineteenth century. In the second part of the twentieth century, a great understanding of aerodynamic and advanced material study helped to return the wind energy extraction which produced electricity well known as wind turbines [13].

The limitation of fossil fuel resources and the environmental, safety hazard of nuclear power, gave a push to the expansion of renewable resources. Wind turbines are promising solutions to many countries worldwide as they do not produce CO_2 emission nor any environmental hazards. However, wind turbines do produce noise, and this has a negative impact on wildlife. To eliminate the negative impact on sites, wind farm setup locations should be selected carefully [14].

To reduce the price and to make energy more competitive, along with the design optimization of rotor blades, wind turbine manufacturers are working on the cost reduction of the wind turbine itself. The production cost of the electrical energy generated includes cost of land, availability of wind at a particular site and interest rates. The production cost is a sum of invested capital and the discounted maintenance capital divided by the discounted production in kWh for a duration of 20 years [15]. However, these production costs can be recovered over a period of 6 to 12 months with a large-scale wind turbine [16].

Many devices convert the kinetic energy which is available in the free stream air into mechanical work which led to researchers proposing unconventional concepts which were not practical and did not reach expectations [17]. A large number of studies took place to increase the power production which returned with few numbers of rewards because of the design limitations and practical effectiveness [4][18][19]. The design limitation is subjected to wind rotor, but it is not the only component that contributes to energy production. A typical wind turbine includes gearbox, control system,

variety of auxiliary units and generator etc, which play a major part in the overall electricity production [13].

The first classification is based on axis of rotation and shaft orientation as shown in Figure 2.1.1. The shaft of horizontal axis wind turbines (HAWT) is horizontally parallel to the ground whereas, shafts normal to the ground are called as vertical axis wind turbines (VAWT) [20].

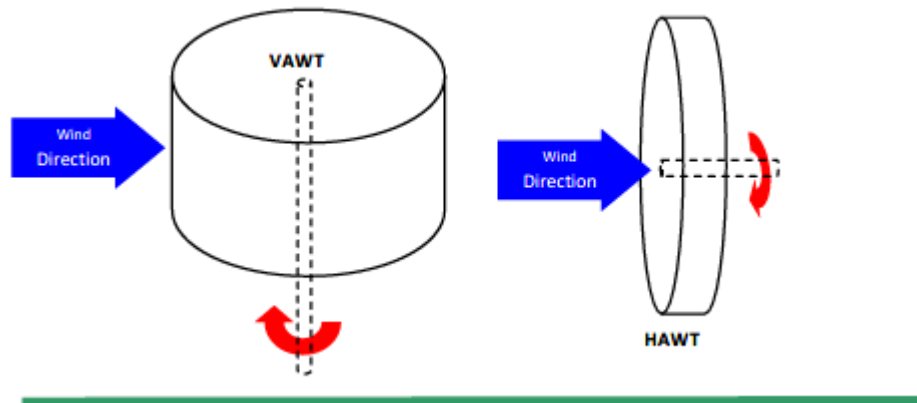


Figure 2.1.1: Shaft and rotor orientation configurations [18]

Factors such as low tip speed ratio, fluctuating RPM and incapability of self-starting of VAWT reduced the focus on its further development [21]. However, tower loads have been reduced by placing the large generator on the ground and is not in need of additional apparatus to face the wind, which gave the boost to undertake further development. VAWT concept has yet to be tested on a megawatt scale, and it will take several years to develop before it can be deemed competitive. The success of the HAWT can be linked to enhanced rotor control through pitch and yaw control, in addition to the challenges associated with other designs. This has caused the current leading large scale wind turbine manufacturers and industrialists to be more inclined to the development of HAWT [18].

2.2 Principles of Wind Energy Conversion

The main component of a wind turbine is an energy converter that transforms kinetic energy in the free stream wind into mechanical energy. Between 1922-25 Albert Betz published a manuscript where this is explained, based on application of elementary physical law. The author explained that only a certain portion of the wind energy can be extracted in the form of mechanical energy when the wind flows through a cross-sectional area [13]. Betz also concluded that the optimal power can be extracted only at a certain velocity ratio between the flow velocities upstream and downstream of the energy convertor.

Betz' momentum theory (BMT) is a study of a one-dimensional plane turbine model with an incompressible, uniformly distributed flow having a linear velocity. When the velocity is linear the rotational flow will not have wake rotation produced by the high lift to drag ratio from the selected aerofoils, tip losses and low tip speed ratio [22,23]. These assumptions will give a rough idea in practical engineering applications, for the calculation of efficiency. From the BMT the maximum efficiency achieved is limited to 59.3%. For wind turbines, the efficiency is termed as power coefficient C_p and is indicated as $C_p = 0.593$ [24]. This overestimated efficiency will have an overestimated turbine torque and power. But in real scenarios the turbine blades will face tip losses and wake effects hence, the maximum theoretical efficiency will not be achieved [25].

2.2.1 The Betz Elementary Momentum Theory

The kinetic energy E is defined as

$$E = \frac{1}{2} \dot{m} v_1^2 \quad (2.1)$$

Where,

$\dot{m}$: mass flow rate of the air (kg/s)

v_1 : free stream velocity (m/s)

The free stream velocity v_1 flowing through cross-sectional area A (m^2) of the rotor for a fixed time unit is known as 'volume flow' $\dot{V}$ (m^3/s).

$$\dot{V} = v_1 A \quad (2.2)$$

The mass flow rate $\dot{m}$ (kg/s) accounts for the air density ρ (kg/m³) and is defined as

$$\dot{m} = \rho v_1 A \quad (2.3)$$

Therefore, the power P (W) can be calculated as

$$P = \frac{1}{2} \rho v_1^3 A \quad (2.4)$$

A 100% energy extraction out of the free stream airflow is not possible. According to Betz' law the wind turbine is at an idealized condition hence there are no losses included in the theoretical calculations. As the power is extracted at an active plane of the wind turbine. The mass flow rate is constant fore and aft the energy converter or the wind turbine blade. But the velocity aft the converter is retarded. To maintain the mass flow rate as constant, the cross-sectional area must be increased [13].

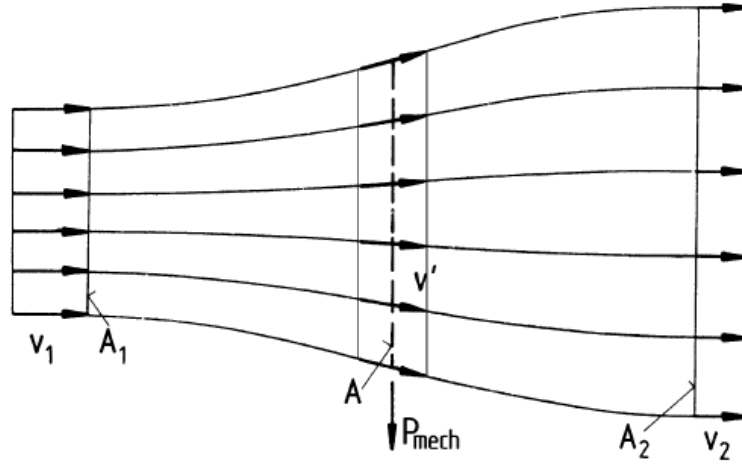


Figure 2.2.1: Free stream air flow according to Betz wind turbine [18]

Figure 2.2.1 shows a disk-shaped converter inside a divergent tube with a free stream air and the mass flow fore and aft the converter. The homogeneous air flow v_1 is in-front of the converter which is then decelerated to v_2 aft the converter and the power Equation (2.4) is expressed as

$$P = \frac{1}{2} \rho A_1 v_1^3 - \frac{1}{2} \rho A_2 v_2^3 = \frac{1}{2} \rho (A_1 v_1^3 - A_2 v_2^3) \quad (2.5)$$

Maintaining constant the mass flow rate and using continuity equation

$$\rho A_1 v_1 = \rho A_2 v_2 \quad (2.6)$$

When the flow is incompressible, the density ρ remain constant and the extracted power is the difference between upstream and downstream energy values, i.e.

$$P = \frac{1}{2} \dot{m} (v_1^2 - v_2^2) \quad (2.7)$$

From the formula to achieve the maximum power, the exit velocity, or the downstream velocity v_2 must be zero. This means that the converter should get the velocity behind it to zero which is physically impossible. It is impossible to achieve zero exit velocity unless the inlet velocity is zero. This means that there is no flow through the converter. If $v_1 = v_2$ then there is no power extraction. So, to achieve the maximum efficiency there must be a value between $v_2 = 0$ and $v_1 = v_2$. To determine this value, the velocity v' should be at a known location, and this is at the converter. Hence the power can be written as $P = F v'$. Where, $F = \dot{m}(v_1 - v_2)$ and by equating power equations

$$\frac{1}{2} \dot{m} (v_1^2 - v_2^2) = \dot{m}(v_1 - v_2)v' \quad (2.8)$$

Therefore,

$$v' = (v_1 + v_2) / 2 \quad (2.9)$$

Thus, the mechanical power output is,

$$P = \frac{1}{4} \rho A (v_1^2 - v_2^2) (v_1 + v_2) \quad (2.10)$$

The power of the wind P_0 without power extraction from the converter through the cross-sectional area A is expressed as

$$P_0 = \frac{1}{2} \rho v_1^3 A \quad (2.11)$$

The ratio between the converter extracted power and the power of wind is known as the power coefficient C_P

$$C_P = \frac{P}{P_0} = \frac{1}{2} \left[1 - \left(\frac{v_2}{v_1} \right)^2 \right] \left[1 + \frac{v_2}{v_1} \right] \quad (2.12)$$

Figure 2.2.2 shows the velocity ratio and the C_P fore and aft the converter. From the plot the maximum ideal power coefficient $C_P = 0.593$ is achieved at $\frac{v_2}{v_1} = \frac{1}{3}$ or ~ 0.33 . By substituting the $\frac{v_2}{v_1} = \frac{1}{3}$ into Equation (2.12) and recalculating the C_P would results in

$$C_P = \frac{16}{27} = 0.593 \quad (2.13)$$

However, these equations were first derived by Betz and the result is known as the “Betz factor.”

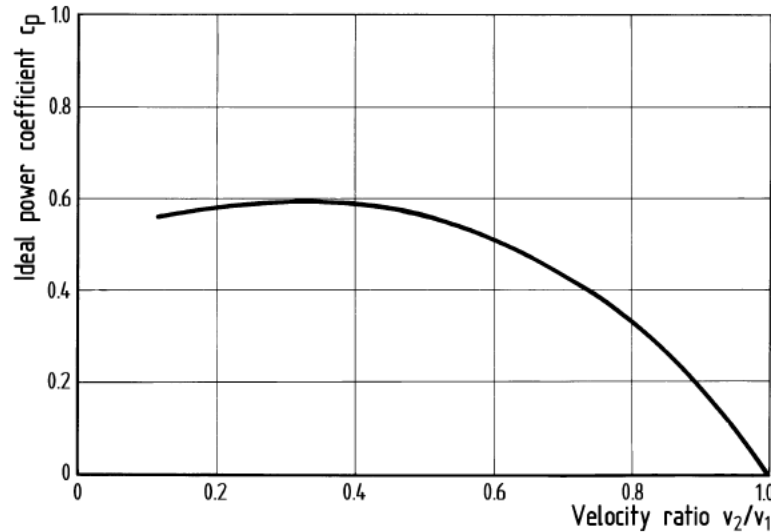


Figure 2.2.2: C_P versus $\frac{v_2}{v_1}$ where v_2 is the downstream velocity and v_1 is the upstream velocity [13]

Figure 2.2.3 shows the detailed flow physics of a wind energy converter along with velocity and static pressure variation fore and aft the disk-shaped converter. The velocity before the converter slowly reduces and is further reduced as it moves behind the converter. The velocity then increases as it reaches the outlet to the maximum diameter. Meanwhile, the static pressure will accelerate till it reaches the converter and will suddenly drop to a low value. The static pressure aft the converter will gradually accelerate back to ambient pressure because of pressure equalisation.

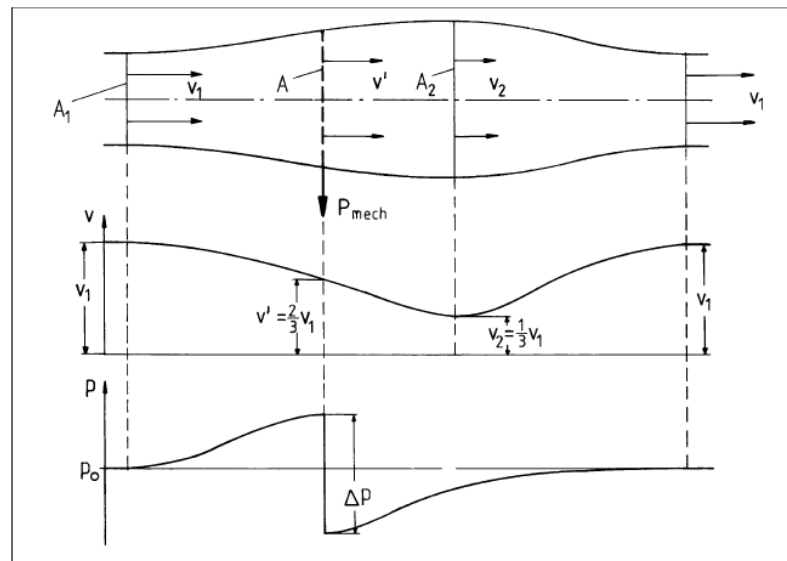


Figure 2.2.3: Ideal disk-shaped energy converter depicting flow conditions while maximum power is being extracted [13]

2.2.2 Power Calculation Using Aerodynamic Lift or Drag

From the Betz momentum theory, the maximum mechanical power was calculated from the free stream velocity at ideal conditions. However, Betz did not consider the effects due to the design of the converter, and it was assumed that the converter was a disk-shaped device; its design and performance characteristics did not contribute to the mechanical power extraction. But it is understood that for real engineering cases, this scenario is not correct. The power extraction from the complete system is dependent of the characteristics of the blade design.

The free stream velocity passes through the entire wind turbine. Hence, the blade produces aerodynamic forces such as lift and drag. The drag force acts in the flow direction whereas, the lift force acts perpendicular to the flow. The power coefficient will purely depend on which aerodynamic force (lift or drag) is being used to calculate the C_p [13].

Rotors Using Aerodynamic Lift

A higher efficiency can be achieved using the aerodynamic lift force. The pressure difference in the aerofoils of the rotor blade help to increase C_p in a HAWT. Figure 2.2.4 is a pictorial representation illustrating the pressure distribution and aerodynamic forces acting on an aerofoil. In the modern era most wind turbines rotor blades are designed using lift configuration. Many theories put forth have been studied to find how lift force is generated. The free stream velocity over an aerofoil will generate the skin friction and pressure difference which in return will produce the aerodynamic forces [26]. These forces are then resolved to get lift and drag and its direction. Further resolving the lift force, lift torque and lift thrust can be achieved. The lift torque will be acting on the plane of rotation which is responsible for the rotation of the blade and the lift thrust will be acting perpendicular to it and will produce the rotor blade thrust [13].

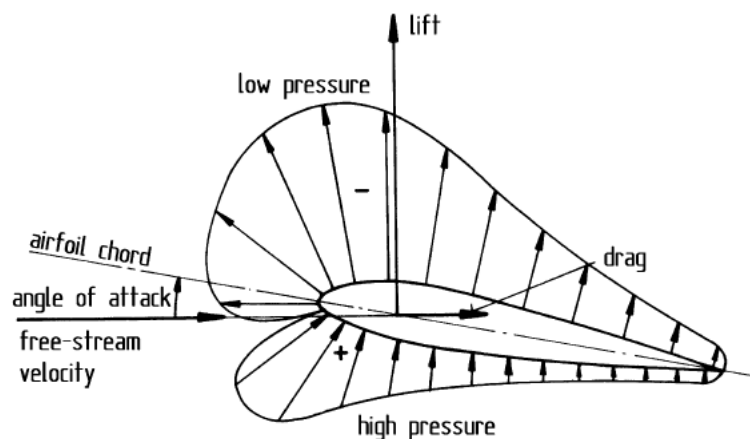


Figure 2.2.4: Aerodynamic forces on an aerofoil [13]

Rotors Using Aerodynamic Drag

Comparing to the lift, the drag force easily converts the wind energy. The mechanical power P can be calculated using the aerodynamic drag D and the velocity v_2 . The drag force D is produced when a velocity v_1 is passing over an area A as shown in Figure 2.2.5.

$$P = D v_2 \quad (2.14)$$

Using the drag coefficient C_D and the relative velocity $v_{rel} = v_1 - v_2$ the drag equation can be written as,

$$D = C_D \frac{\rho}{2} (v_1 - v_2)^2 A \quad (2.15)$$

Hence the resultant power from Equation (2.14) can be expressed as

$$P = C_D \frac{\rho}{2} (v_1 - v_2)^2 A v_2 \quad (2.16)$$

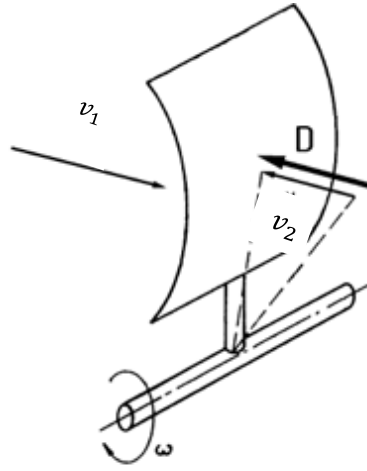


Figure 2.2.5: Representation of drag device experiencing aerodynamic forces and flow condition [13]

From Figure 2.2.2 in Section 2.2.1, the maximum efficiency is achieved at velocity ratio of $\frac{v_2}{v_1} = \frac{1}{3}$. By resubstituting Equation (2.16) into Equation (2.12) the maximum power coefficient will be

$$C_{Pmax} = \frac{4}{27} C_D \quad (2.17)$$

And the maximum C_D for a concave surface designed wind turbine is 1.3 [13].

Hence,

$$C_{Pmax} \approx 0.2 \quad (2.18)$$

2.2.3 Comparison of Rotors Using Aerodynamic Lift and Drag

To confirm the maximum power coefficient produced by two types of converters after a thorough investigation, Smeaton (1759) for Dutch smock mills a $C_{Pmax} = 0.28$, realized that the power coefficient could be increased to $C_{Pmax} = 0.50$ using modern high lift devices [23].

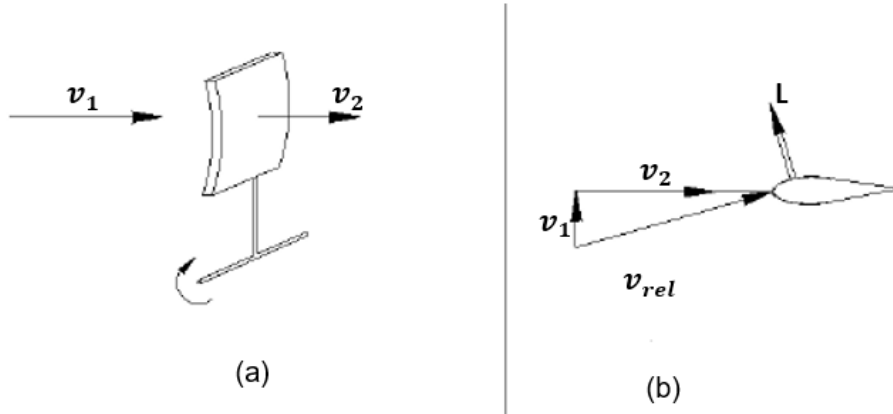


Figure 2.2.6: Comparison between (a) drag and (b) lift driven rotor [23]

Drag and lift equations in terms of relative velocity is represented as

$$D = \frac{\rho}{2} v_{rel}^2 s C_D \quad (2.19)$$

$$L = \frac{\rho}{2} v_{rel}^2 s C_L \quad (2.20)$$

The fundamental difference between the drag and lift driven rotor is due to the relative velocity, as shown in Figure 2.2.6. In a drag driven wind turbine the relative velocity, v_{rel} is calculated as

$$v_{rel} = v_1 - v_2 = v_1(1 - \lambda) \quad (2.21)$$

Where,

$$\lambda = \frac{v_2}{v_1} \quad (2.22)$$

In case of drag driven rotor the relative velocity v_{rel} is always less than free stream velocity v_1 because of $v_1 - v_2$.

Whereas, in a lift driven wind turbine

$$v_{rel} = \sqrt{v_1^2 + v_2^2} = v_1 \sqrt{1 + \lambda^2} \quad (2.23)$$

From Equation (2.23) the relative velocity v_{rel} of the lift driven rotor is always higher than free stream velocity v_1 .

Where,

v_1 : free stream velocity (m/s)

v_2 : angular velocity (m/s)

λ : tip speed ratio

$$\lambda = \frac{\text{Blade tip speed}}{\text{wind speed}} \quad (2.24)$$

For a drag driven wind turbine $\lambda < 1$ and for a lift driven wind turbine $\lambda = 1$ to 15. Therefore, the tip speed ratio value will determine the multiple of free stream velocity i.e., relative velocity.

When both lift and drag driven wind turbines have the same swept area s , the lift driven rotor's aerodynamic force magnitude is directly proportional to the relative velocity squared, hence the force attained by the lift force driven rotor is a multiple of the force attained by the drag force driven rotors [23]. The “active

rotor plane” obtained by the drag driven wind turbine is very small as compared to the Betz efficiency. However, 59.3% of Betz efficiency cannot be achieved even in lift driven wind turbine because of the losses that has been neglected in Betz efficiency principle [27].

2.2.4 Aerofoil Theory

Betz’ theory is based on a disk-shaped converter to extract the power. However, currently, an aerofoil structured rotor blade is implemented in place of a converter to extract the power and thrust. Aerodynamic forces produced by the aerofoils will give additional efficiency. A symmetrical aerofoil in a free stream velocity v_1 at angle of attack (AOA) $\alpha = 0^\circ$ with no lift (L), but with drag (D). So, the symmetrical aerofoils need to have some AOA to produce lift and drag as shown in Figure 2.2.7.

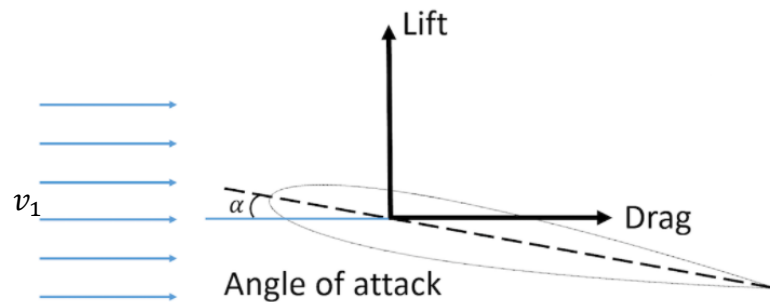


Figure 2.2.7: Diagram illustrating lift, drag, and angle of attack on NACA 0012

The aerodynamic forces are expressed as

$$L = \frac{1}{2} \rho v_1^2 C_L C \quad (2.25)$$

And,

$$D = \frac{1}{2} \rho v_1^2 C_D C \quad (2.26)$$

Where,

C : chord length (m)

C_L, C_D : lift and drag coefficient respectively

The Reynolds number (R_e) is the ratio of “inertia force acting on a unit volume of the fluid”. Viscous force (μ) is indirectly proportional to (R_e), hence if the viscous force is higher the Reynolds number will be lower and vice versa.

$$R_e = \rho v_1 C / \mu \quad (2.27)$$

For a NACA 0012, the lift coefficient C_L linearly increases up to AOA 10° which is $C_{L_{max}}$ and above this AOA the flow separates, and stall occurs at a Reynolds number of 50,000. Meanwhile, C_D increases with the AOA. The difference in pressure between the upper and lower surfaces of an aerofoil determines the lift. Hence using a cambered aerofoil will be beneficial because it produces more than zero lift coefficient at AOA zero. However, when considering a horizontal large-scale wind turbine, both aerodynamic and structural characteristics need to be considered [28]. When considering the inner part of the rotor blade, the aerofoils need to be thick enough to support the blade structurally, and the outer part of the blade is responsible for the aerodynamic efficiency L/D . To achieve a better wind turbine performance lift to drag ratio need to be as high as possible [9].

2.2.5 Velocity Triangles

In an upwind turbine, the incoming wind of velocity v_1 will pass through the pressure surface of the rotor blade, this velocity v_1 will drive the turbine to rotate and the rotating turbine will produce an induced or relative velocity v_{rel} . When a blade rotates with an angular speed Ω a local circumferential speed $v_2 = \Omega r$ will be present in the axis of rotational plane. As shown in Figure 2.2.8, the

angle of relative wind φ is a sum of the pitch angle θ and the angle of attack α , i.e.

$$\varphi = \theta + \alpha \quad (2.28)$$

Introducing the tip speed ratio λ_r at cut radius.

$$v_2 = \Omega r \quad (2.29)$$

$$\lambda_r = \frac{v_2}{v_1} = \frac{\Omega r}{v_1} \quad (2.30)$$

The direction of φ with reference to the rotor plane is

$$\tan \varphi = \frac{v_1(1-a)}{\Omega r (1+a')} = \frac{(1-a)}{(1+a')\lambda_r} \quad (2.31)$$

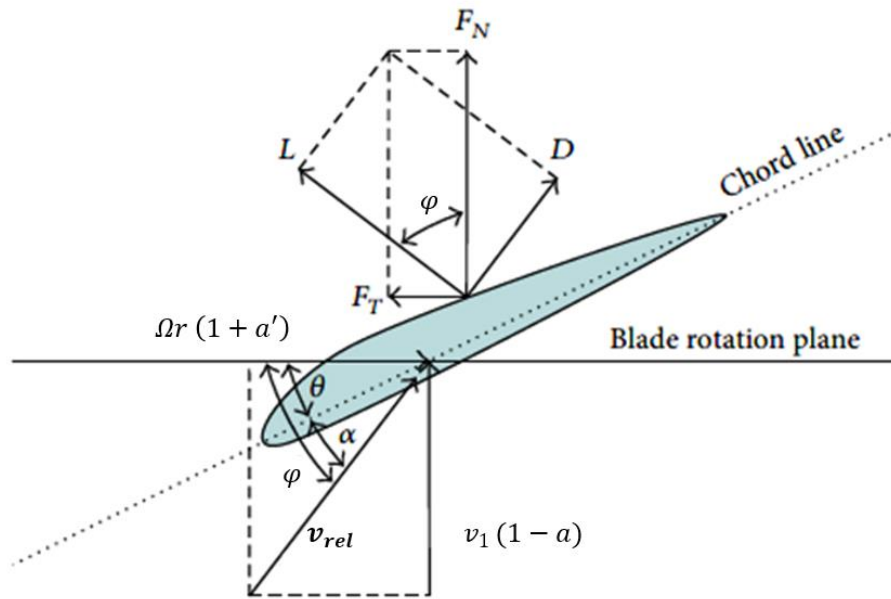


Figure 2.2.8: Velocity triangle of wind turbine aerofoil cut section [7]

From the Figure 2.2.8, substituting $a = \frac{1}{3}$ and $a' = 0$

$$v_{rel} = \frac{v_1(1-a)}{\sin \varphi} = \frac{2v_1}{3\sin \varphi} \quad (2.32)$$

Therefore,

$$\tan \varphi = \frac{2}{3\lambda_r} \quad (2.33)$$

$$\varphi = \tan^{-1} \left(\frac{2}{3\lambda_r} \right) \quad (2.34)$$

From Equation (2.34), the angle of relative velocity φ at each radial cut r can be calculated.

2.2.6 Wind

Understanding basic concepts of wind is vital before designing a wind turbine. The amount of energy from wind is proportional to wind speed. This means that a wind farm must be selected very carefully to meet the required efficiency. The distinct characteristic of the wind is variable both temporally and geographically. Even though it is possible to predict the seasonal variation of the wind, still on a shorter timescale there will always be a large variation [29].

Turbulence caused by thermals, land features and weather will vary the wind every second. Due to the surface friction the flows are stronger above the ground level than closer to the ground as shown in the Figure 2.2.9.

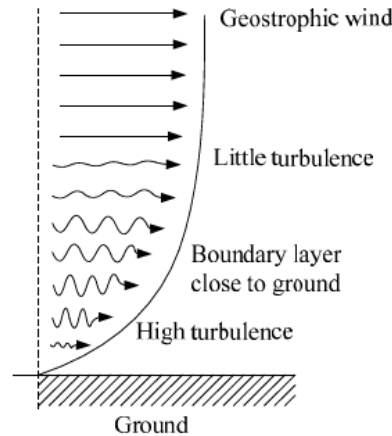


Figure 2.2.9: Atmospheric boundary layer [23]

This results in variable loads being acted on the blades of the wind turbines when rotating, hence the design of the blade must withstand environmental conditions, which are rarely constant.

While extracting power, the turbine has a mutual impact on the wind itself. When the wind reaches a blade, its velocity reduces. Further reduction in wind speed is seen when the wind is in contact with the blade, hence, forming an imaginary circle around the blade's tips, that is called the swept area. This ultimately reduces the available power. Some of the wind heading towards a blade gets diverted due to the slow-moving air in its vicinity which results in the turbine not being able to capture optimal power. To avoid all these issues, it is better to reduce the upwind speed by one third to reduce the wind speed, allowing maximum energy capture, which is 59% of the wind power (Betz' law). However, the maximum energy being captured through current designs is about 40 to 50% [30].

2.3 Rotor Aerodynamics

The rotor is the primary functional component of a wind turbine. The entire system is under the influence of the rotor properties, such as aerodynamic, dynamic, and overall cost standpoint. The rotor blades transform the wind energy into useful mechanical energy, and the efficiency of energy conversion determines the effectiveness of wind turbine. Wind flowing through the swept area of a rotor is a fluctuating power input which is then converted to uniform torque with the aid of aerodynamic and dynamic properties induced in the rotor in addition to reducing the unavoidable dynamic loads.

Maximum efficiency can be achieved with an upwind rotor and three blades. Few of the two bladed wind turbines are designed for a downwind rotor. An efficiency increment of 3% can be achieved with a three bladed design when compared to a two bladed design. However, only 0.5% efficiency increase is obtained with three to four rotor blade designs. Hence, a three rotor blade is preferred [19].

2.3.1 Blade Element Momentum Theory

According to Betz's momentum theory [13] a 2D flow over actuator disc and the streamlines are in one plane. Whereas the rotor will further transmit

rotational motion which generates the rotor wake. The angular momentum is preserved when the spin within the wake is inversely corresponding to the rotor torque. Due to rotating wake produced by extended momentum theory the turbine blade power coefficient will always be less than Betz value. Therefore, the power coefficient is dependent on the ratio of translational and rotating motion of the air stream and is known as tip speed ratio λ .

The blade element or strip theory is an interaction between the rotor blade aerodynamic characteristics and its shape [31]. According to the blade element theory, the aerodynamic force and upwind condition act at a distance r from the axis of rotation, assuming zero interface in aerodynamic forces. An aerofoil at distance r is placed with a pitch angle θ with reference to axis of rotation as shown in Figure 2.3.1.

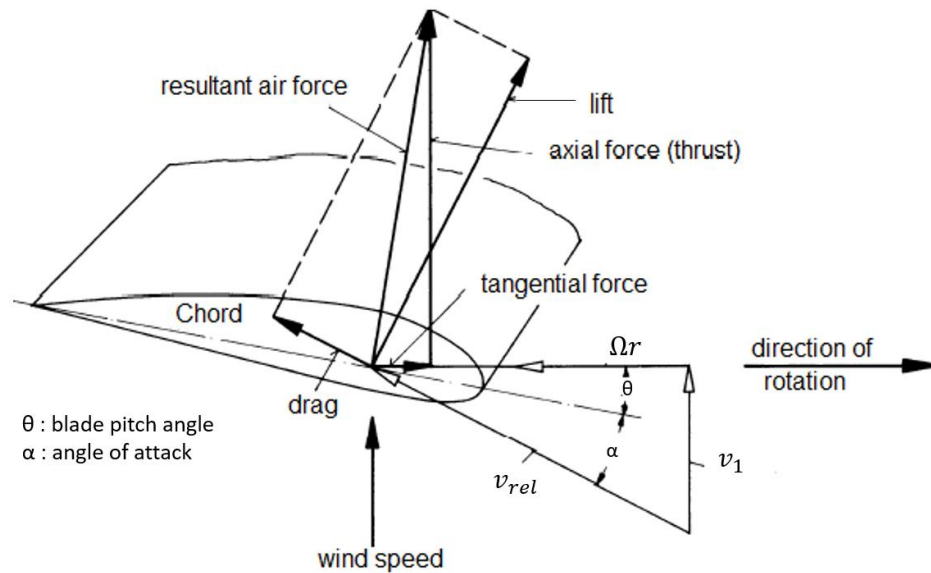


Figure 2.3.1: Aerofoil cross section of a blade element experiencing dynamic forces and velocities [13]

2.3.2 Rotor Power Characteristics

Wind rotor's power characteristics are predominantly influenced by fluctuation of its power with changing wind velocity. Torque and thrust characteristics are also significant. The rotor thrust, and torque characteristics are also critical. A rotor's power characteristics can be represented dimensionless and it is function of the rotor power coefficient of the tip-speed

ratio [13]. These quantities determine the shape and size of rotor blade and wind turbine component's assembly.

2.3.3 Power and Torque Characteristics

As a result of the simple momentum theory, “the order of magnitude of the mechanical power output of the rotor” can be determined. The aerodynamic rotor theory describes how a rotor configuration's geometrical shape is interrelated to its power characteristics. A rotor power coefficient C_{PR} is used to calculate the rotor power P_R , based on the velocity.

$$P_R = C_{PR} \frac{\rho}{2} v_1^3 s \quad (2.35)$$

Where,

s : Swept area (m^2)

v_1 : Velocity of the wind (m/s)

C_{PR} : Power coefficient of the rotor

ρ : Density of the air (kg/m^3)

According to the strip theory, the power coefficient C_{PR} is calculated for a given rotor speed to wind speed ratio, or a given tip speed ratio (TSR) [13]. Variation of power coefficient with TSR can be obtained by repeating strip theory for various tip speed ratios [13].

Power coefficient of rotor can be calculated either at fixed rotor speed for various wind speed or vice versa [32]. Power coefficient curves for every individual pitch angle of blade utilized in operation must be determined when the rotor has a blade pitch control (Figure 2.3.2).

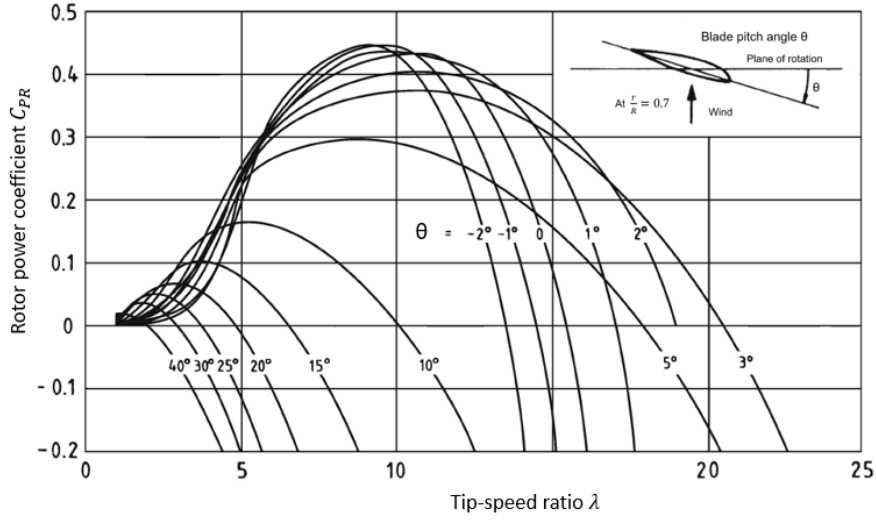


Figure 2.3.2: Power characteristics of experimental WKA-60 wind turbine rotor [13]

The torque M (Nm) is the other important defining parameter. The torque coefficient C_{MR} , is used to calculate the rotor torque

$$M = C_{MR} \frac{\rho}{2} v_2^2 s R \quad (2.36)$$

R is the radius of the rotor, and the rotor torque M is the ratio of power by rotational speed. Therefore, power and torque coefficient can be related as:

$$C_{PR} = \lambda C_{MR} \quad (2.37)$$

2.3.4 Aerodynamic Power Control

The power captured by the rotor at high wind speeds is immensely higher than the boundaries imposed by the rotor structural stability. The safety margin of strength limit of large wind turbine is narrowed as the rotor size increases. However, the maximum power capacity of a generator is also a factor limiting power output of a rotor blade.

Speed limitation on a rotor blade is vital due to several reasons. For example, when there is a power outage, the generator loses torque abruptly which will further cause the rotor speed to increase rapidly. If there were no aerodynamically effective countermeasure in place which would react immediately, the turbine could end up being destroyed.

Changing the aerodynamic angle of attack or the reduction in projected swept area of the rotor, or varying the effective free-stream velocity at the rotor blade, will influence the reduction of driving aerodynamic forces [13]. However, the effective free-stream velocity at rotor blades only changes with rotor speed because wind speed cannot be controlled [13]. In a variable-speed operating wind turbine, the power control can be achieved through the implementation of the rotor speed as a corrective or limiting variable [13]. Altering the rotor speed can be considered a last backup option due to its contribution to power range being very small. Only very small rotors can reduce the aerodynamically effective swept area of rotors by turning it out of the wind (furling) [13][33].

2.3.5 Power Output Prediction

A wind turbine's output changes with free stream velocity, and each individual velocity will have its own power performance curve as shown in Figure 2.3.3. The energy produced by a wind turbine can be estimated using a power performance curve neglecting the technical characteristics of its many components [34]. Power curves describe outputs of “electrical power as a function of hub height wind speed”.

Three points on the wind turbine performance curves are as follows: the least wind speed at which the machine can generate sufficient power is called as “cut-in speed”. The wind speed at which maximum power output is obtained is known as “rated wind speed” and the highest wind speed at which a turbine can generate power is termed as “cut-out speed” [34].

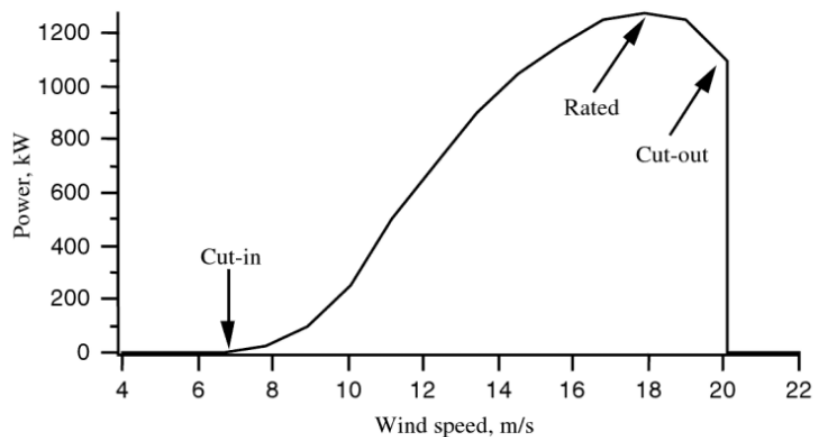


Figure 2.3.3: Wind turbine power curve [34]

2.3.6 Power Control by Rotor Blade Pitching

Mechanically altering the rotor blade pitch angle is an efficient technique for modifying the aerodynamic angle of attack, and consequently the power input. The rotor blade is generally twisted along its longitudinal direction with help from actively controlled actuators for this purpose [35]. There have also been attempts to use the influence of centrifugal forces to obtain passive pitch control. As shown in Figure 2.3.4, and in theory, there are two ways to adjust power by modifying the rotor's aerodynamic angle of attack. To reduce power input, the traditional solution is to lower the angle of attack of the blade. The angle of attack of a blade is directly proportional to the power input hence increasing the angle of attack will increase the power input. Increasing the blade pitch angle until it reaches "critical aerodynamic angle of attack" is an alternative method, when the flow separates at the rotor blade surface, further restricting the aerodynamic power input, which is called "stall". Benefits from following this procedure as such that it requires a lower turning angle to effectively pitch the blade [8].

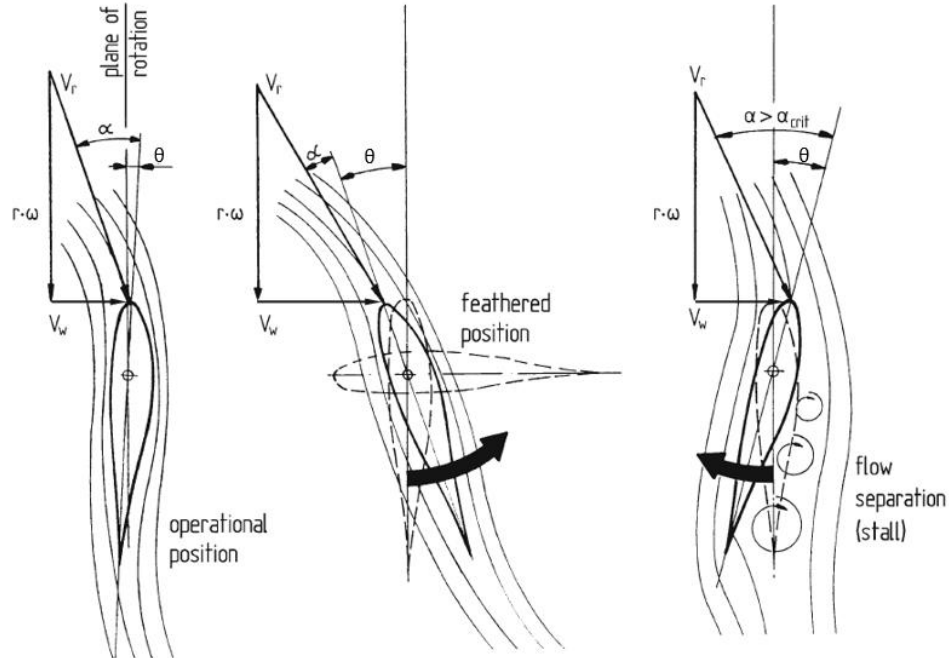


Figure 2.3.4: Feather or stall pitching to control the rotor input power [13]

2.3.7 Aerodynamic Characteristics of an Aerofoil

The aerodynamic characteristics of the aerofoils are utilised to influence the efficiency of rapidly rotating wind rotors to a significant extent. Choosing the most appropriate aerofoil and obtaining a smooth rotor blade surface are indeed crucial to rotor blade performance and manufacturing. This discovery, which is well-known in aviation, has recently obtained recognition in wind turbine technology. Along with rotor performance, control and operating behaviours are also influenced by aerofoil characteristics. As the aerofoil crosses the critical angle of attack the stall occurs due to a sudden drop in lift curve. This nature of the aerofoil is also important due to the rotor blade being subject to different angle of attack when operating [36].

The lift to drag ratio L/D is one of the most important aerodynamic properties of an aerofoil. The lift is 200 times greater than the drag for contemporary aerofoils [13]. Hence lift driven wind turbines which has higher L/D aerofoils will have superior performance.

$$C_L/C_D = E \quad (2.38)$$

Lower lift to drag ratio will lead to reduced power coefficient hence the design tip speed ratio λ will also be reduced. C_{PR} is influenced by L/D and λ . The C_{PR} has very little impact due to number of rotor blades when L/D and λ are high, otherwise if L/D and λ are low, the number of blades becomes important. Hence, a low-speed rotor needs multiple blades, although the aerofoil characteristics of these blades are not important. However, high speed rotors perform well with fewer blades, but the aerodynamic characteristics of the aerofoils used in these blades determines the power generated [18].

2.3.8 Boundary Layer

A slow-moving fluid will have a thin boundary layer near the body surface where the viscous force is predominant, therefore the flow separates at higher Reynolds number.

From the stagnation point, the boundary layer starts to grow in thickness. Till the distance l from the stagnation point the boundary layer of the fluid is smooth and called 'laminar' flow. After distance l the Reynolds number is called 'critical Reynolds number' $Re_{crit} = \rho v_1 l / \mu$ and then the 'turbulent' flow starts (Figure 2.3.5). Due to turbulence, the fast-moving fluid will mix with the boundary layer which then delays the point of separation [22].

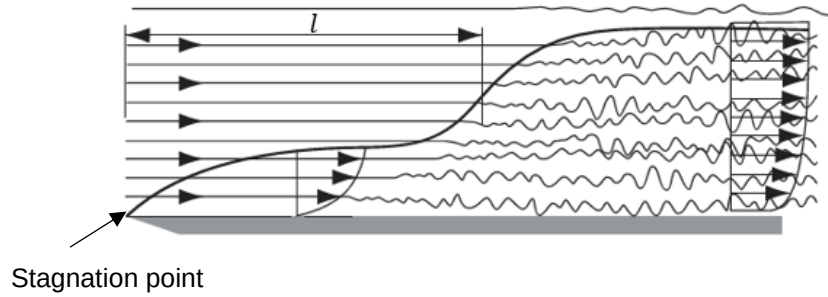


Figure 2.3.5: Laminar and turbulent boundary layers [22]

2.4 Blade Design for HAWT

HAWT is a popular option in the wind turbine industry due to its enhanced performance. The wind turbine will have impacts if the blade design and its profile changes. Hence, an extensive study is required to understand the parameters of blade profile that influences the performance of a wind turbine.

2.4.1 Tip Speed Ratio (TSR)

The TSR (λ) is the ratio of rotor speed to relative wind speed. All other rotor dimensions can be derived from this design parameter. When selecting the TSR, several parameters such as torque, efficiency, noise, mechanical stress, and aerodynamics are taken into consideration, these parameters having a direct impact when a change of TSR occurs. For Instance, when the TSR increases, the efficiency of the wind turbine drastically increases [23]. However, when considering the impacts of increased centrifugal stress, aerodynamic stress, and high noise levels the increase in efficiency is not significant. Increase in TSR causes reduction in blade chord widths further leading to narrow blade profiles. Although, for narrow blades material usage for

manufacturing would be less further reducing production cost. The higher tip speeds are related with increased centrifugal and aerodynamic forces. This increased force makes it difficult for the structural stability to be preserved and prevent blade failure. The tip speed ratio design consideration and its impacts are shown in Table 2.4.1.

Table 2.4.1: Tip speed ratio design implications [18]

Tip Speed Ratio	Low	High
Value	Tip speeds of one to two are considered low	Tip Speeds higher than 10 are considered high
Utilisation	traditional windmills and water pumps	Mainly single or two bladed prototypes
Torque	Increases	Decreases
Efficiency	Decreases significantly below five due to rotational wake created by high torque	Insignificant increases after eight
Centrifugal Stress	Decreases	Increases as a square of rotational velocity
Aerodynamic Stress	Decreases	Increases proportionally with rotational velocity
Area of Solidity	Increases, multiple 20+ blades required	Decreases significantly
Blade Profile	Large	Significantly Narrow
Aerodynamics	Simple	Critical
Noise	Increases to the 6th power approximately	

When the TSR is increased, the aerodynamics of the blade becomes growingly important. The blade design for higher relative wind speed will develop less torque at low velocity, which will lead to high cut-in speed with demanding self-starting [37]. The level of noise rises to around the sixth power. [23][38]. Maximum energy is achieved in modern wind turbines when the TSR for a three-rotor blade is six to nine, and for two-rotor blade it is nine to ten.

2.4.2 Shape and Quantity of Blade Plans

BEM is used to achieve an ideal blade shape, this is done by calculating chord length in accordance with Betz' limit, lift generated by the aerofoil and local air velocities. There are different theories to calculate the ideal chord length however they range in complexity [13][23][37][39]. Theoretical foundations are estimated by Betz' optimization Equation (2.37), and good approximation can be derived using aerofoil cross-section with tip losses and minimum drag for rotor blades with TSR of six to nine [3]. In case of low tip speeds this method is considered inaccurate while using an aerofoil with high drag and blade section near the hub. Drag losses and wake should be considered for such instances [23][39]. Using Betz' method a wind turbine basic shape can be defined, but more complex optimization methods are frequently employed in practise. Figure 2.4.1 shows the regional classification of a wind turbine blade.

$$C_{opt} = \frac{2\pi r}{B} \frac{8}{9C_L} \frac{v_{wd}}{\lambda v_{rel}} \quad (2.39)$$

Where,

$$v_{rel} = \sqrt{v_2^2 + v_1^2} \quad (2.40)$$

- C_{opt} : optimum chord length (m)
- r : radius (m)
- B : number of blades
- v_{rel} : local relative velocity (m/s)
- C_L : coefficient of lift
- v_1 : free stream velocity (m/s)
- v_2 : angular velocity (m/s)
- λ : local tip speed ratio
- v_{wd} : design wind speed (m/s)

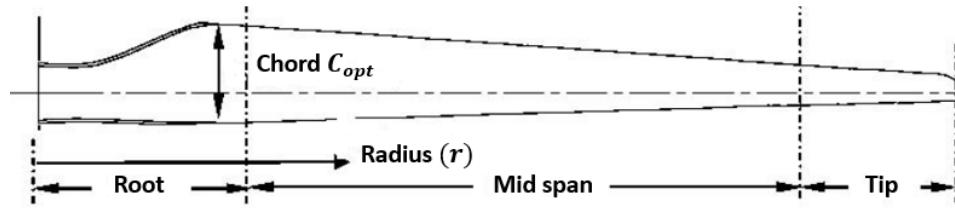


Figure 2.4.1: Blade form representation with regional classification [18]

To identify the optimum chord length using Equation (2.39), the number of blades is considered negligible in connection with efficiency. Nonetheless, in reality considering blade losses, a 3% loss occurs for a 2 bladed design and 7%-13% blade loss occurs for a single bladed design in comparison with three bladed designs [22]. The efficiency offered by a four bladed design is minor hence it does not justify the additional cost incurred for manufacturing and material used for the extra blade. The dynamic load will increase with the number of blades used [40] when deciding a suitable number of blades, the tower load should also be taken into consideration [22]. The three blade wind turbines are not just aesthetically pleasing they do tend to have smoother blade rotation and are visibly well ordered in stationary position, when compared to one or two bladed design which tends to have an obvious jerky motion [13].

2.4.3 Configuration

The use of fewer blades, results in a significant decrease in manufacturing costs and weight of the rotor nacelle. Polar asymmetrical rotor will have obvious balancing difficulties and dynamic structural complications [40]. One and two bladed rotors are also associated with greater wear, poor aesthetic features, and bird conservation issues [41][42]. Environmental, commercial, and economic requirements are well achieved by a three-bladed wind turbine design; hence, it is the most efficient configuration, which is widely used in the current large-scale industries. Figure 2.4.2 shows the modern large scale wind turbine configuration

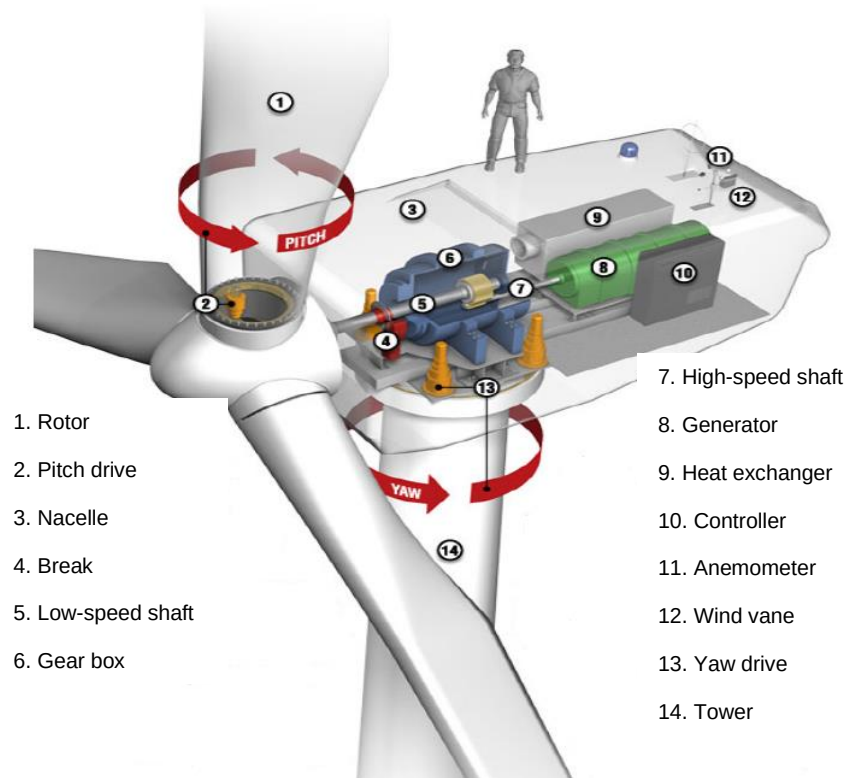


Figure 2.4.2: Modern large-scale wind turbine configuration [18]

2.4.4 Aerodynamics

For an effective wind turbine blade design, characteristics of aerodynamic performance are essential. [43]. The lift force is accountable for the generation of the turbine's power; hence it is crucial to increase the lift force by an improved design of the blade. A resistive drag force is generated due to friction, which will resist the motion of the blade, and therefore should be reduced. The choice of an aerofoil section with a high L/D , generally higher than 30 [43], should be chosen.

While openly available software, such as XFOIL [44] model results accurately excluding high angles of attack, aerofoil thickness and post stall circumstances, lift and drag coefficients of aerofoils are difficult to predict numerically. At various Reynolds numbers and angles of attack, the experimental validation of typical aerofoils is correlated using lift and drag tables.

To design wind turbine blade aerofoils, aircraft technology has been used traditionally, including similar cross-section thicknesses and Reynolds

numbers, which is appropriate for blade tip conditions. While arriving at a final design for a wind turbine blade, additional attention needs to be given to aerofoil profiles as this will directly impact the mechanical loads and operating conditions [45].

Wind turbine blades have structural limitations that need the usage of high thickness to chord ratio aerofoils near the hub. Within the aircraft sector, these aerofoils are considered ineffective. Lift-to-drag ratio of thick aerofoil sections will be often lower. Since thick aerofoils are used in wind turbines for their structural stability, a special interest arises to increase the lift of such aerofoils to make their usage efficient [45][46]. RISO [47], LS, SERI-NRE, FFA [22], and Delft University [45], are the aerofoils which are specially designed for wind turbines and are available to meet the demands of the wind turbine industry.

It would be inefficient to use a single aerofoil throughout the span of the blade [43]. Structural needs and relative velocities vary from hub to tip of the blade; hence, aerofoils must be selected according to the corresponding design point. The blade sections at the root will be carrying the maximum load of the blade, hence, an aerofoil with thick profile must be selected. However, the aerodynamic performance of this aerofoil must satisfy the basic requirements. Crucial aerodynamic performance with less load and faster linear velocity is faced closer to the tip of the blade as aerofoils gets narrower. The aerofoil thickness to chord ratio needs to be greater than 27% at the root, 27 – 21% at mid span and 21 – 15% at the tip of the blade.

2.4.5 Angle of Twist

The lift generated on an aerofoil is mainly due to the angle at which the aerofoil cross-section is inclined with respect to the freestream air. This angle is called angle of attack. At a given radius, the RPM and wind velocity establish the air stream inflow angle.

The required angle of twist is established, based on the angle of attack and TSR. The wind turbine blade tends to have high wind speed to radial velocity ratio, which is the main reason for the orientation of the aerofoil of the blade at the hub to be more inclined towards the wind flow (inwards) and the tip of the blade remains mostly normal to the flow.

Blade twist angle can be further reduced, to lower the manufacturing cost of the wind turbine. This might cause the aerofoil to operate in lower optimum angle of attack, further causing lower lift to drag ratio. An adequate justification needs to be provided for these changes as they also result in low turbine performance.

2.4.6 Blade Thickness

When selecting the thickness of a rotor blade, there will always be a trade-off between “aerodynamic efficiency and rotor blade stiffness and strength requirements”. From an aerodynamic point of view, a wind turbine blade needs to be as thin as possible, so that high-performance aerofoils can be used. The load-bearing components, on the other hand, must have an appropriate thick cross-section to meet structural specifications. Spar height will determine the maximum thickness and is a critical parameter for satisfying stiffness requirements while maintaining structural weight minimal. Such two criteria, appropriate rigidity, and efficient aerodynamics are mutually exclusive. The only viable solution is to balance the structural weight penalty in contrary to the variations in power obtained. As a result, aerodynamic designer's responsibility is limited to demonstrating the impact of the power output and thickness-to-chord ratio on rotor efficiency [13].

2.4.7 Off-Design Conditions and Power Regulation

In olden days, wind turbine blades were designed to rotate at fixed rotational speed due to the gearbox technology and generator in place, which led to an inefficient result at different tip speed ratio. However, at the rated wind speed the required results were obtained. [13]. This is no longer true for the current larger turbines, and the gearbox might become unnecessary in future turbines [36]. Fixed-speed turbines are now only used in smaller designs, and so the corresponding issues faced because of the off-design can be ignored.

To obtain optimum dimensions for a wind turbine from the design wind speed, the actual wind measurements from the installation site is used. But wind conditions vary with location, and the turbine must work in off-design situations, for instance for higher-than-rated wind speeds. To avoid massive loading of the

hub, blade, generator, and gearbox, a means of controlling rotational speed must be adopted. The turbine must also retain a high performance even when the wind velocity is well below the rated wind velocity.

Since the angle of incidence of the resulting airflow upon the blade is directly affected by the approaching wind velocity, the pitch angle of the blade should be adjusted correspondingly which is referred as pitching because it keeps the aerofoil section's lift force constant. To change the blade angle, the entire blade span is mechanically rotated from the hub. This approach is useful for improving lift in situations where the velocity is below the rated velocity and for preventing rotor overspeed, which can result in severe blade failure at high loads, or generator overload [13].

At high wind speeds, the lift force can be minimised by pitching the blade in two different ways which affects rotor's rotational velocity. First, the angle of attack can be decreased by reducing the angle of pitch, hence, a lower lift is produced. This technique is termed as Feathering. Or a stall condition can be achieved at maximum angle of attack at its critical limit by increasing the pitch angle and decreasing the lift, as a substitution. When pitching the blade, the feathering needs the highest number of mechanical movements. Stalling, on the other hand, might result in high dynamic loads, though it is still preferred. Around the blade, the uncertain transition from "attached to detached airflow" results in excessive dynamic loads which further leads to unwanted fluttering [13].

A rotor blade can be designed for a limiting wind speed using the stall condition which is called "passive stall control". At high wind speed and rotor speed, an angle generated which results in stall, reducing rotor speed automatically. In fact, guaranteeing that stall happens correctly is challenging, hence a safety margin is frequently used. Because of a safety margin utilization, the performance of the blade is well below the designed operational conditions, hence this methodology can only be used in small scale wind turbines [48].

Recent wind turbine industries employ feathered pitching to the span of the blade at hub. Performance, flexibility, and ability to pitch the blades completely to a resting position by implementing the feathered pitching. According to reports, manufacturers use collective pitch [49], which means that all the blades are pitched at the same degree. Pitching blades separately, on the other hand,

can result in even more load reductions [50]. Most designs do not require an additional mechanism, and it is predicted to be extensively used [48][50].

2.4.8 Smart Blade Design

"Smart Blades" is a new blade design concept being studied in the current research, where the shape of the blade is changed based on wind conditions. Various techniques, such as smart actuator materials or aerodynamic control surfaces are included in this area of blade design. Barlas [32] has given a detailed review about this topic. The goal of this study is to reduce fatigue loads and ultimate (extreme) loads while simultaneously enhancing the extraction of dynamic energy. Numerous wind energy research institutes are investigating the subject, which is mostly inspired by principles of helicopter control.

Series of aerofoils are used to build an efficient wind turbine rotor blade that are combined and twisted together and end in a circular flange [23][51] as shown in Figure 2.4.3. Loss reducing tip geometries can be included. Various optimizations could be made to make production easier:

- Decreasing the twist angle.
- Linearizing the chord width.
- Reducing the number of different aerofoil profiles.

All manufacturing simplifications are unfavourable for an efficient rotor blade and are better to avoid unless necessary. Development of modern moulding materials and techniques have enabled the production of increasingly complicated blade designs. But any ultimate geometry is dictated by the costs of manufacturing and the difficulties of the design analysis. Major optimisation elements, like twist angle, variable chord length, and numerous aerofoil designs, are now included in major wind turbine providers' products.

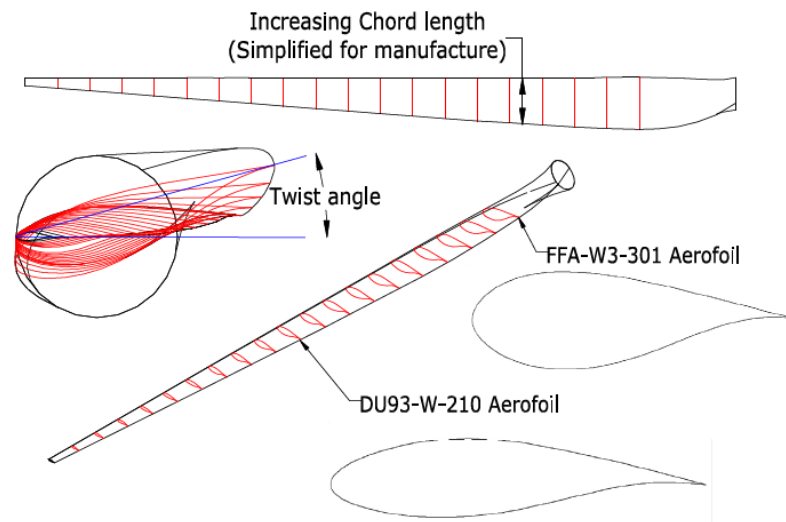


Figure 2.4.3: A current HAWT blade featuring an increased linear chord length, various aerofoil shapes, and twist angle [18]

2.4.9 Control Methods

Control methods or techniques employed in wind turbine control systems are vital as they do not just reduce maintenance cost and ensure efficient performance, they can also be used to ensure safe operation, structural stability/life span of the wind turbine and to obtain optimized power output. This is ultimately controlled by two factors:

- Rotational speed of the wind turbine
- Generator capacity or speed

The power can be limited to or optimized by use of control methods. The wind turbine can be controlled by generator speed, rotation of the wind turbine and blade angle adjustment.

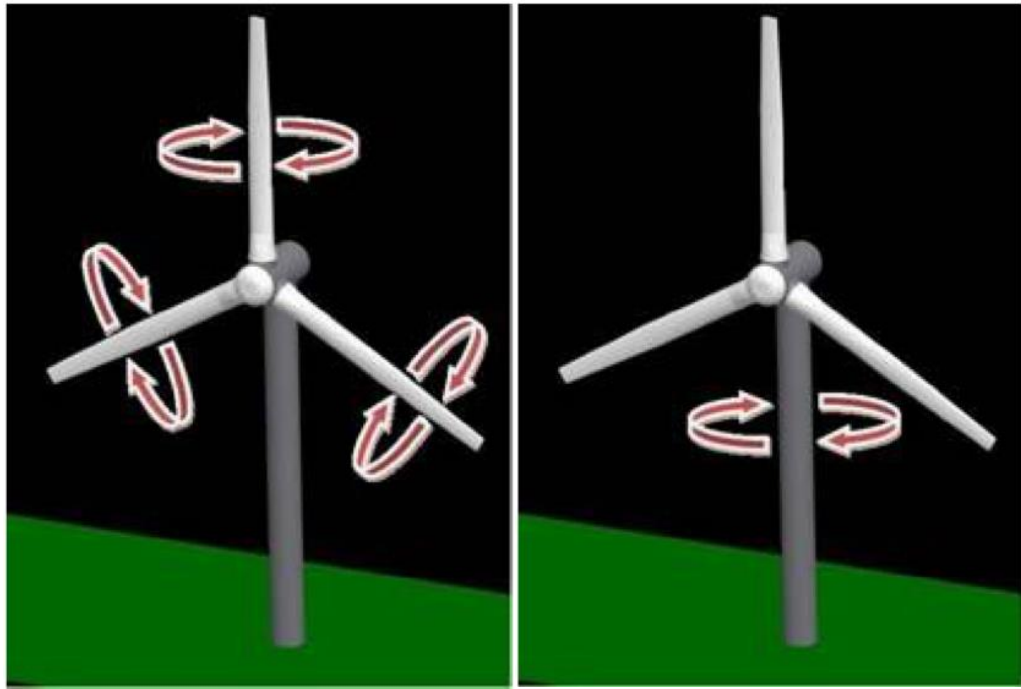


Figure 2.4.4: Pitch (left), yaw (right) adjustment [52]

The blade angle adjustment is termed as pitch control and the rotation of the whole wind turbine is termed as yaw control as shown in Figure 2.4.4. Certain rotor speeds and power output can be achieved by using pitch control. This adjustment can be made to make the blade experience stall and furl. Stall is a condition where the blade flat side further faces into the wind which in-turn increases the angle of attack. Furling will cause the edge of the blade to face the wind also causing a decrease in angle of attack. In a wind turbine operating at high wind speeds, pitch control is considered to be a very efficient and effective method to limit power output as it changes the aerodynamic force acting around the blade [52].

The blade gets aligned to the wind flow direction when a yaw control method is in use. This does not just increase the wind turbine's majority of rotor area being exposed to the oncoming wind, this also reduces power losses due to wind flow directional changes. The turbine, if static without yaw control, might not be facing the direction of wind flow, this will result in power losses.

ΔP power losses can be estimated by

$$\Delta P = \alpha \cos(\varepsilon) \quad (2.41)$$

Where, ε is the yaw error angle.

The final type of control system is the one that makes use of electrical sub-systems in the wind turbine. The generators are normally coupled with power electronics namely electronic convertors. Stator and rotor present in the generators are the two types of generator control systems. The stator being the stationary part and the rotor is the moving part at certain time. If one of them is disconnected from the grid a change in synchronous speed of the generator can be observed, this change is independent of frequency or voltage of the grid. Hence, this control method can be concluded as one of the most effective methods at low wind speeds.

2.4.10 Control Strategies

Pitch and generator control method are the most effective methods to optimize the power output of a wind turbine. “Fixed-speed fixed-pitch, fixed-speed variable-pitch, variable-speed fixed-pitch, and variable-speed variable-pitch” are developed based on the pitch and generator control method to have a control over the turbine functions in all parts of the power curve, variable-speed variable-pitch being the ideal curve [52]. Figure 2.4.5 shows the power curves for all four strategies developed.

Fixed Speed - Fixed Pitch (FS-FP): In this control strategy both speed and pitch are fixed throughout the power curve. The performance of the turbine cannot be improved with this configuration, the generator speed locks the power line frequency hence causing the rotational speed to be fixed as well. This strategy is not ideal because the power output drops at higher wind speeds (after rated wind speed is reached). To not exceed the rated power output, the ratio of the gearbox becomes an important selection factor.

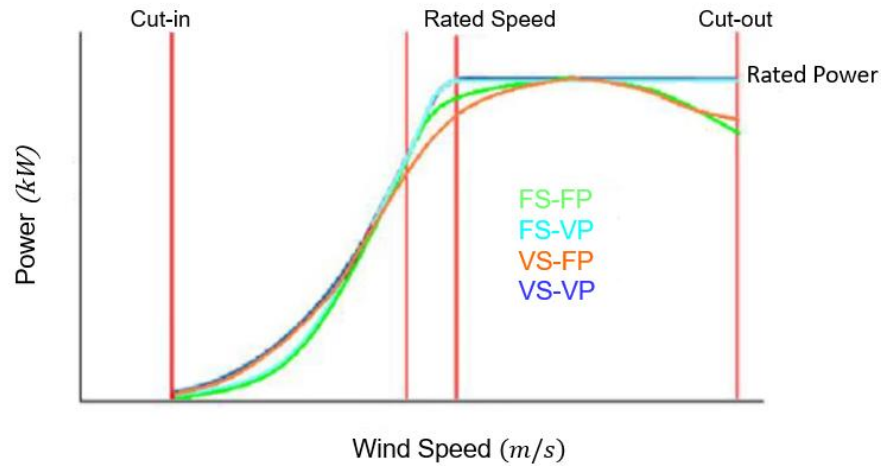


Figure 2.4.5: power curves for various control strategies (VS-VP, being the ideal curve) [52]

Fixed Speed - Variable Pitch (FS-VP): In this control strategy a maximum power output is obtained at a fixed speed, however varying pitch ensures to extract the maximum power at any wind speeds, further assuring no power output losses. Also, the wind turbine will have a fixed pitch below the rated speed and variable pitch after the rated speed. Although the power output remains constant after the rated speed is reached.

To limit the power output, feathering and stalling can be employed with this configuration. For feathering to take place huge amount of control design factors are consumed meanwhile the stall will produce additional thrust which would in-turn damage the tower. Hence the pitch must be varied carefully to avoid the unwanted losses.

Variable Speed – Fixed Pitch (VS-FP): In this control strategy the generator electronically controls the synchronous speed which in-turn varies the rotational speed of the rotor based on the wind speed that the wind turbine is subjected to. The generator's rotor and the drive train are independent of the grid and are free to rotate. Hence the generator drives the rotations of the rotor blade. This concludes the generator being independent of the grid frequency. A fixed pitch completely depends on the blade designed to limit the power, through no more generation of lift when the wind speed increases, which is termed as passive stalling.

Variable Speed - Variable Pitch (VS-VP): This strategy is the combination of VS-FP and FS-VP to bring out the best out of the two existing configurations. Hence below the rated speed VS-FP occurs to maximize the power out, ensuring efficient operation and quality. Above the rated wind speeds FS-VP configuration is utilized to achieve the maximum power output without any losses until the operation reaches the cut-out speed. In conclusion VS-VP is the only ideal power curve that is theoretically achieved.

2.6 Critical Review of Literature

The first large wind turbine was manufactured in 1941 which produced 1.25 MW electricity. An offshore wind farm was first built in 1991 in Denmark. The water depth of the offshore wind farms was between two to five meters. Eleven wind turbines were placed on this farm and could power around 2200 households and after 25 years the wind turbines were dismantled. In 2013 the first floating offshore wind turbines were invented, producing 20 kW, based on a concrete-composite floating platform at a depth of 45 meters or more [13]. The present work is carried out to build a large offshore wind turbine blade which can produce higher power, is cost-effective, and reliable.

The specifications of the aerofoil were determined using the blade element method and the computational fluid dynamics at 3 stall regions of the wind turbine. Either method could be used for the stall correction. The first step for this method is to find the local angle of attack followed by the local twist angle and rotational speed, where the local flow angle has been considered to be zero. The axial velocity plot is used to determine the axial induction factor. The EllipSys3D code has been used for the rotor design simulation, it is a representation of incompressible, Reynolds average Navier-stokes (RANS) equations [53].

The k- ω SST turbulence model is used to perform steady and unsteady simulations. The Blade Element momentum code uses an empirical formula for high axial induction factors. The tip effect has been corrected using the Prandtl's tip losses factor with no stall delay condition. The Tellus [54] and Nordtank [55]

wind turbines have been considered for a local time-stepping algorithm and a fully turbulent flow was considered with the simple traditional model. The mechanical power plot showed a good agreement between CFD and BEM results with the assumption without using empirical stall conditions as shown in Figure 2.6.1 [56].

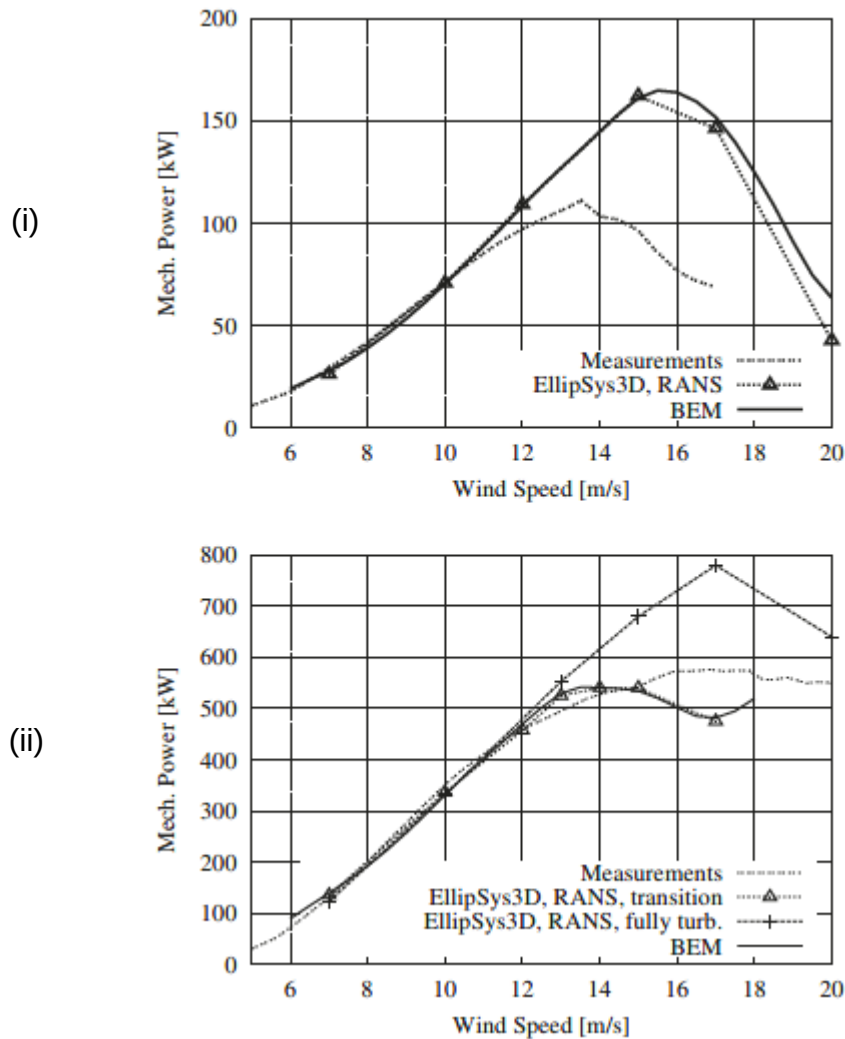


Figure 2.6.1: Mechanical Power versus Wind speed curves for (i) Tells and (ii) Nordtank [56]

CFD-based results have an acceptable agreement compared to BEM's ones near the mid-span and having small variation at the root and the tip region of the rotor, which concludes that the results from CFD can be utilised to calibrate induction modelling for BEM tools. Flow separation and yawed conditions cannot be studied in the BEM theory because this approach is 1D. However, CFD can account for them [57].

Rahimi [58] used several methods such as inverse BEM, Average azimuthal technique, the Three-Point, Shen, and the Ferreira-Micallef methods, to calculate the different angles of attack and the induced velocity. Results from each method vary due to the input conditions such as the monitoring point location, and the turbulence wake correction. The chord is a good monitoring point location since it varies from root to tip. At mid-span, a good agreement was shown between all the selected methods in terms of axial induction factor, but there was a small variable at the root and tip of the WT blade. This variation is due to the assumption of circulation along the span, the 3-D effects are dominant at the root and tip of the rotor blade with an influence of the induced velocity and can be fully captured through a 3-D CFD study.

This study also gives a clear idea about the circulation along the span and the flow separation at the hub due to radial flow. In the axial direction the faraway monitoring point location will not have the same upstream and downstream this is also a reason for the variation of the results at the root and tip of the rotor blade [59].

The thickness of the aerofoils varying from 15% to 40% of the chord, has been studied using XFOIL versions. A modified version of XFOIL, RFOIL, was used to improve the aerodynamic parameters [60]. In the current era, the DU aerofoils are being used globally for different rotor diameter varying from 29 *m* to 100 *m*, with a maximum power up to 3.5 *MW*. A study was conducted to compare results of XFOIL, RFOIL and wind tunnel-based C_L results of DU aerofoils. The results showed a very good agreement, although there was a slight variation at high angles of attack (at $C_{L\ max}$), due to the turbulence model considered. RFOIL had convergence problems, even though its result almost matched the $C_{L\ max}$ results of the wind tunnel [45].

Research states that the outboard aerofoil must not leave a large difference between designs C_L and $C_{L\ max}$ to avoid the loads in case of gust and the difference must not be too small to avoid the stalling of the rotor while the controller is too slow. A maximum difference of 0.2 is sufficient in such design cases. A high $C_{L\ max}$ becomes desirable in the case of stall-controlled turbine

as it results in smaller chords and reduced loads. However, high $C_{L\max}$ design generally encounters an abrupt stall [45]. The inboard aerofoil must have a significant thickness. A high thickness at the root is required to withstand the structural load of the blade, the expected thickness of the aerofoil is expected to be between 30% and 40% of the chord, and the inboard aerofoil needs to have higher C_L to achieve sufficient torque, even though a lower wind-speed is present at the root [61].

Large offshore wind turbine blades need to sustain high loads, these loads are balanced by the aerofoil structures. One of the aims is to design an aerofoil from the family of aerofoils to withstand the loads at high Reynolds number, which could also produce or maintain the power output. The inviscid panel code XFOIL was selected to study the aerodynamics and structural performance of the selected aerofoils. The relative thickness of the aerofoils were varied (18%, 21%, and 25%) and the comparison was carried out through CFD, wind tunnel tests, and Wind Multi Block (WMB) methods [62].

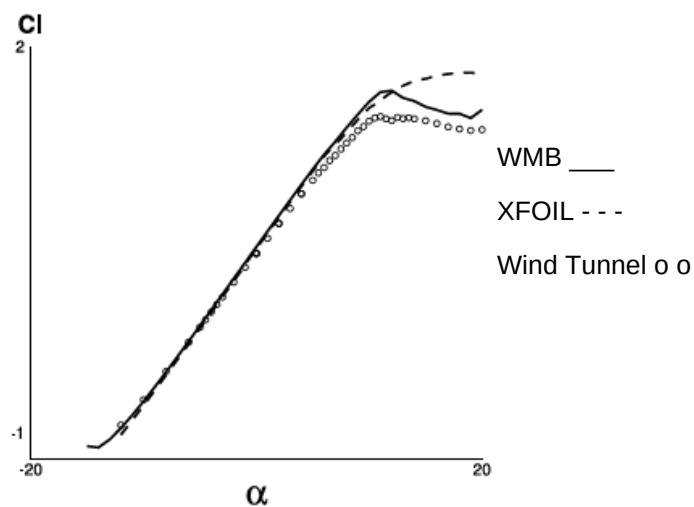


Figure 2.6.2: Lift coefficient for an aerofoil with 18% thickness, at $R_e = 9 * 10^6$ [62]

The redesign of these aerofoils was made by considering the structural design and reduction of load. The aerofoil shape was described using Bezier curves (fourth and fifth curves) [63]. The aerofoil with 18%, 21% and 25% thickness were placed at tip mid-span and at the root respectively to achieve better efficiency and lift to drag ratio. The characteristic of each aerofoil was

studied and compared with CFD and WMB. Only the 18% thickness aerofoil was manufactured for wind tunnel testing. The WMB and XFOIL results were validated against experimental results obtained from wind tunnel testing, see Figure 2.6.2 [62].

From an aerodynamic point of view, a thin aerofoil with a high lift to drag ratio is preferred, structural points need a thick aerofoil which have high elastic strength, while manufacturers need a non-varying aerofoil. However, the focus in this PhD work, is on the aerodynamic design. It is important to achieve a high lift to drag ratio at the blade tip, hence the aerofoil must be thin. At the root, the blade must be structurally strong hence a thick aerofoil is selected [64].

Two new aerofoil designs for 10 – 20 MW offshore WT were investigated to get a lightweight rotor; they have also been demonstrated to be suitable regarding a structural point of view. The aerofoil near the hub (rotor root) needs to be thick to give a structural stability. However, the aerodynamic performance at this section will not affect the overall aerodynamic performance of the blade. Hence, to improve the structural characteristics, a thick and blunt trailing edge aerofoil has been designed using XFOIL [65].

Clean and contaminated conditions were considered to investigate the turbulence effect on the aerodynamic torque. The aimed results of the structural and aerodynamic behaviour have been checked with the Blade Element Momentum theory. The aerodynamic efficiency is improved when the aerofoil has a high lift coefficient and a low drag coefficient, but decreasing the drag is quite difficult because the shape of the blade is blunt. Hence, the lift coefficient of the rotor blade must be improved, keeping the point of structural requirement. In the first design, the thickness of the aerofoil was 50% and the lift coefficient was increased by 56%. However, the drag coefficient was high, and the overall aerodynamic efficiency was dropped by 30%. The second design was for 40% thickness. In this case, the lift coefficient was increased by 21% whereas, 60% of the overall performance was reduced [66].

The maximum power efficiency $C_p = 0.593$ can be achieved from the Betz limit, and a constant linear velocity profile has been assumed in this theory. The tip losses reduce the efficiency due to wake rotation or turbulence. The low tip

speed increases the wake rotation which in-turn reduces the efficiency. At high tip speeds, high lift to drag ratio aerofoil, and special tip geometries can help to improve the efficiency [67]. Betz' momentum theory yields an acceptable estimation for three blades wind turbines in the modern horizontal axis wind turbine (HAWT), which utilizes 6 to 9 tip speed ratios with negligible drag. Assumptions made are that the lift coefficient to be maintained, the principal design of a blade should be dependent of the number of blades and tip speed, which reduce the usage of material hence, provide cost reduction [68].

Increasing the number of blades increases the efficiency but does not provide a rationale for the manufacturing cost of an extra blade. This additional load on the tower also is an added extra cost. Hence three-bladed design with yaw and pitch are accepted across the globe. The aerodynamic parameters such as width, thickness and twist angle are key to obtain a better efficiency. In the present era manufacturers are looking for cost-effective blade design through increasing the rotor size rather than a minimal increase in efficiency. This motivates researchers to design large wind turbines with higher efficiency [69].

Hosseini [70] has simulated the accuracy of the NREL Phase VI wind turbine through CFD Modelling. A large 2 MW WT has been selected and the scaled-down model has been numerically simulated. The 'scale factor' has been used for the variable properties of the WT. The scale factor states that criteria such as number of blades, blade profile, and tip speed ratio, have to be achieved to have a scaled-down model for the numerical analysis [71]. For large and small WTs, the flow conditions must be the same considering the relative velocity angle obtained from the velocity triangle.

The results (Figure 2.6.3) of the simulation have shown a good agreement with a 15% error at 7m/s, this is due to the real-time flow pattern and the turbulence model of the scaled-down WT. The Reynolds number at the rotor blade root for a small and large wind turbine is different because of the turbulent flow.

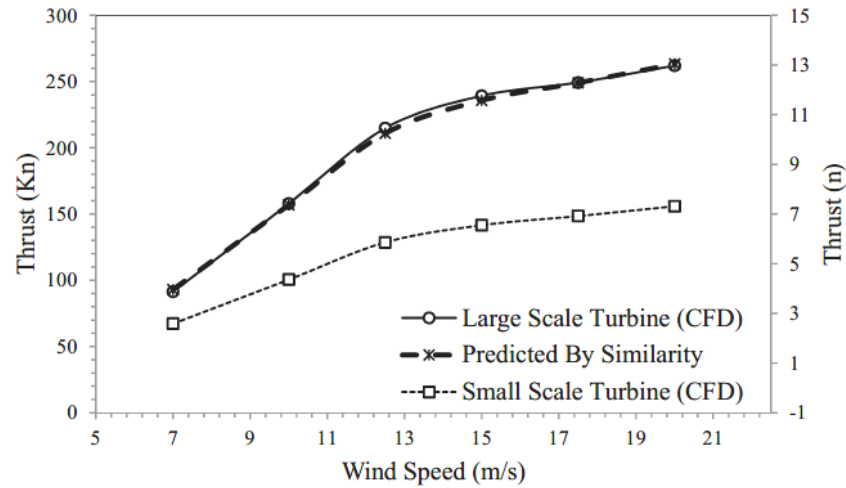


Figure 2.6.3: Thrust versus wind speed result comparison [70]

The tip effects caused by downwind velocity have been reduced by the attachment of winglets. As compared to upwind winglets, downwind winglets are more efficient, according to C_p optimization studies [72]. The shorter downwind winglet ($> 2\%$) produces less power than the increased radius of the rotor blade. Wind tunnel results showed a 27% increase in power coefficient ($C_p = 0.45$) at a tip speed ratio four as compared to the WT without winglet. But when the tip speed ratio is increased to eight the results were dropped. Hence, a horizontal axis WT with a winglet cannot achieve better aerodynamic efficiency at higher tip speed ratio [72].

The reduction in axial velocity due to the circulation at the tip of the WT blade in the axial direction is negative and the radial flow is positive due to mass conservation, henceforth the power produced by the downwind winglet is negative. An upwind winglet produces positive power in the main rotor blade by reducing the axial velocity. However, the winglet does not produce additional thrust which will affect the loading at the blade tip [2].

The momentum theory provoked a mismatch with the CFD code results, and the downwind winglet was better in comparison to the upwind winglet regarding power augmentation. This mismatch was because of the neglected effect of tangential induction momentum theory; hence the coefficient of thrust was higher for an upwind winglet which increased the power coefficient.

However, in the CFD code EllipSys3D study, the power coefficient showed a downwind winglet as efficient as compared to upwind ones [73].

Various studies [74][75][76] have shown the improved aerodynamic performance of winglets on a nonrotating wing. Farhan [77] studied the combined effects of winglets and aerofoils on a rotating horizontal axis WT. Numerical tests and optimizations were carried out by using different winglet designs like “heights, cant angle (upward angle of the winglet), planforms and aerofoils.” The NREL phase VI WT was selected as the baseline, elliptical and rectangular winglet planforms were attached to the baseline WT to compare the CFD results with S809 and PSU 94-097 aerofoil profiles. Figure 2.6.4 shows the results of design variants.

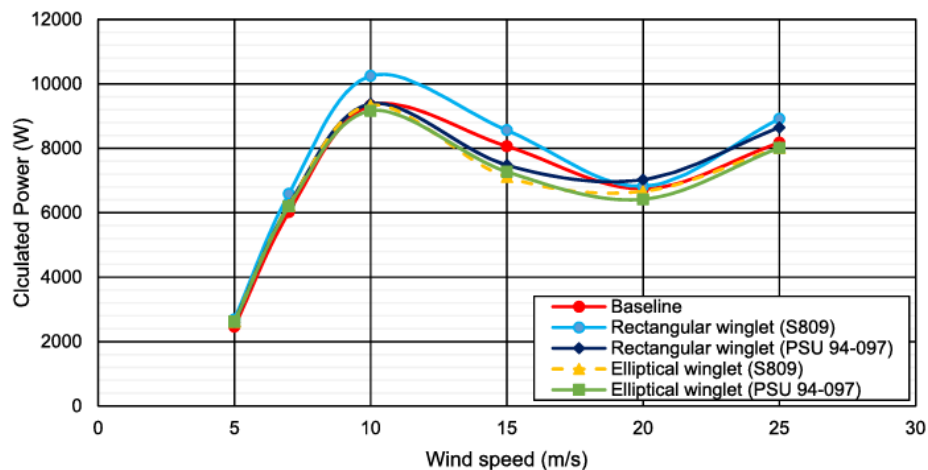


Figure 2.6.4: Calculated power versus wind speed comparison for different winglet design [77]

The NREL phase VI [78] is a two twisted and tapered blade upwind horizontal WT and is built with S809 aerofoils. Wind tunnel results were compared against CFD-based obtained with the $k-\omega$ SST, and Spalart-Allmaras turbulence models. The Spalart-Allmaras' based simulation provided power results that were over-predicted due to the stall at 11 to 25 m/s. However, numerical results from both Spalart-Allmaras and $k-\omega$ SST models agreed well with the wind tunnel results. Only a small discrepancy in the results was present around 9 to 11 m/s, due to the transition flow which takes place at this speed. Hence further studies were carried out using Spalart-Allmaras and $k-\omega$ SST

models. Four configurations were studied with the baseline WT, Rectangular winglet with S809 and PSU 94-097 and elliptical winglet with S809 and PSU 94-097 with cant angle 45° [79]. The results showed that a rectangular winglet with S809 aerofoil has a 9% increase of power at 10 m/s but has a 1% reduction of power at 20 m/s because it is in a stall regime compared to the baseline.

Zahle [80] also studied three different winglet designs along with the chord, twist, and blade length parameters. A surrogated rotor model was created and studied in CFD and numerically winglet designs were optimized concerning different geometry and loads. The results showed an increment in power production by 2.6% with winglets and only 0.76% increment when the length of the rotor blade was extended conventionally. The “IEA-10.0-198 10 MW Reference Wind Turbine” [81] was selected as the baseline and the blade tip was redesigned using 3D CFD. Overall, 12 winglet designs were created by taking account of span wise splines for chord and twist, winglet curvature, winglet offset, and winglet height at velocity of 6, 8, and 10 m/s . The research concluded that the power produced by a winglet design is equivalent to the tip extension of the blade length by 10 % to 15%. The winglet does influence the increment in the coefficient of power however it has a limitation on load constraints. A simple curvature at the blade tip is an added advantage to improve the power by reduced vortex at the tip.

To prevent the tip vortex, a scale-down model was selected according to the laboratory size, different type and shape of winglets were studied to choose the optimum design. The mechanical power was measured for different winglets by exposing to turbulence. These turbulence levels reduced the vortex and the wake formation. The wake and free stream were mixed to reduce the tip losses. The HAWT blade was used to study the interaction among the tip vortex and the free-stream turbulence, where the “Torque Matched Aerodynamic Shape Optimization” (TMASO) method was implemented to design the scale-down model. TMASO can also be used to scale up the blade with approximate error-free results [82].

However, the reduction of tip vortex might increase the C_p but it is not the only reason, there might be other possible reasons for the increment because

of the turbulence intensity level [83]. The energy developed due to the turbulence intensity setting at the inlet might also be a reason for the increment in the C_p , which produces the eddy frequencies in the y/D direction of the blade. The introduction of turbulence level does reduce the tip vortex hence a reduction of tip losses, which in-turn, helps in fast recovery of the wake region by mixing the free stream. The eddy frequency also energizes the wake and reduces adverse pressure effects. Hence, the additional power gained is not only because of the additional winglet but also due to the free-stream turbulence effects.

The "NREL offshore 5 MW baseline wind turbine" was designed by Jonkman [84]. It is an upwind three-blade wind turbine with "variable speed variable blade-pitch-to-feather-controller". The rotor radius of the 5 MW wind turbine is 63 m. To create this model various published documents were collected from the literature. The selected baseline model was the REpower 5M machine [85]. Few of the unavailable data were collected from windPACT [86], RECOFF [87], and DOWEC [88] projects. The main intention of the "NREL offshore 5 MW baseline wind turbine" project was to make the model available for all the industrial and organization researchers' world-wide to use them as the baseline.

Lars [3] designed a "10 MW Offshore Wind Turbine Rotor". The rotor blade was designed aerodynamically as well as structurally. The design wind speed was used to establish the blade diameter. 13.25 m/s was the selected design wind speed, and the obtained rotor diameter was 141 m using aero-elastic simulations. The speed at the tip was predicted to be 90 m/s. The rotor blade was designed using BEM and 47.9% of power coefficient was achieved. Further to the aerodynamic design, the structural design was carried out. It is indeed very important to balance aerodynamics with structural designs.

Bak [12], designed a "10 MW reference wind turbine". A detailed study was carried out on the aero-servo-elastic design, aerodynamic rotor design, and structural design. The blade was designed based on a trial-and-error method. The cube of the rotor radius was directly proportional to the mass of the turbine. The diameter of the rotor is important because of its weight and associated

costs. Hence, the blade was designed by considering the 5 MW wind turbine rotor blade. The blade length was up scaled to 86.37 m. The remaining parameters were up-scaled using classical similarity rules [89].

FFA, DU and Riso aerofoils are the few existing aerofoils that can be used for the rotor blade. However, all the aerofoils' coordinates are not open access, hence, FFA aerofoils [90] were selected for the design. The HAWTOpt2 [91] design tool was used to aerodynamically optimize the blade. Increasing the thickness do reduce the aerodynamic efficiency hence, relative thickness on the blade's outer part was not increased more than 24.1%. 80 m/s was chosen as the maximum tip speed. The structural and aero-servo-elastic design was also carried out.

2.7 Summary of literature review

For the last decades wind energy has been considered to be very important due to green energy being continuously developed. Wind turbine rotor blades were designed using the BEM theory. On small scale horizontal axis wind turbines, a large amount of research has been carried out using BEM, CFD code and experimental methods. Commercial large scale offshore wind turbines now need additional development to improve their capacity to produce a high amount of electricity.

A wide range of studies have been carried out on the aerofoils of rotor blades. The higher the thickness, the better the structural properties, but it does affect the aerodynamic efficiency. Hence, the thickness of the blade is limited to 15% – 21% at blade tip, 21% – 27% at mid span and 27% at the blade root. FFA aerofoils [10] and DU aerofoils are the most preferred aerofoils because of their thickness and lift to drag ratio [28][44][45][46].

The research on reduction of vortex or wake effect and turbulence level showed that designing and optimizing the winglet does not contribute much to the power coefficient. Varying the pitch and flap design of a large wind turbine make the design complex and thus increase the costs. Manufacturers however require a simple design to reduce costs.

A “5 MW Reference Wind Turbine” was designed to overcome all these issues in 2009. Later in 2011 and 2012 a “10 MW Reference Wind Turbine” was designed. The classical similarity rules have been used to up-scale the rotor blades. The key parameter to up-scale the wind turbine is the rotor blade diameter but increasing the diameter will have a negative impact on the mass. A large wind turbine requires a large nacelle, and the added load increases the size of the tower. Hence, the importance of a large offshore wind turbine design is crucial as it must be cost-effective, reliable and it should produce a large amount of power.

3. Methodology

3.1 Introduction

Fossil fuels are used world-wide as the main source of energy. Air pollution, climate change and environmental challenges are caused due to utilization of fossil fuel and they are serious hazard for humans [92][93]. An established alternative for the negative consequences due to the use of fossil fuels, is through the production of clean energy, by saving energy, through improved efficiency, and by using renewable energy instead. Lot of attention is being received for the installation of wide range of renewable energy extraction units [94].

The two most common kinds of wind turbines are horizontal axis wind turbines (HAWTs) and vertical axis wind turbines (VAWTs). HAWTs can produce large amount of power and has been typically considered. Latest evidence has shown that improved VAWTs can also produce equivalent results along with additional benefits, depending upon the cases [95][4].

Air that flows through the wind turbine has a complex pattern and behaviour. Firstly, the atmospheric air is turbulent in nature, the flow field around a wind turbine is highly unstable [96]. Due to the supporting tower, there will be vortex shedding in a large-scale wind turbine. However, near the attached boundary layer, it is produced in finest scales [97]. Secondly, HAWTs which have pitch-regulation will undergo dynamic stall [98]. The dynamic stall is much more complicated than a static stall. Static stall takes place at a critical angle of attack of a stationary blade. However, on the blade suction surface, a large flow is separated due to delay in the flow and vortex shedding, this is known as dynamic stall and is most severe at HAWT blade's inboard region [99][100].

3.2 Turbulent Flow Modelling for Wind Turbines

The Navier-Stokes equations are derived for conservation of momentum which when merged with the continuity equation can determine the flow physics of a Newtonian fluid and an incompressible flow in three dimensions [101]. From the conservation law of energy, an equation for compressible flow can be derived by including the density ρ .

Power output and aerodynamic loads can be estimated through the aerodynamics of the wind turbine. Various degrees of complexity and accuracy of the wind turbine model can be determined by aerodynamics loads [4]. Using the BEM methodology two-dimensional (2D) data of an aerofoil can be obtained for the design phase [102]. To reduce the computational cost, the flow physics of a wind turbine can be studied through three dimensional (3D) inviscid vortex approaches. Navier-Stokes equations are neglected when considering the results of these two approaches. Computational fluid dynamics (CFD) is used to resolve the next stage of the model. The most widely used turbulence models in CFD are Reynolds-averaged Navier-Stokes (RANS) based models.

3.2.1 Reynolds-averaged Navier-Stokes equations for CFD

Uniform and accurate flow physics around a wind turbine can be numerically determined in CFD using Navier-Stokes equations as compared to BEM and vortex methods. Large Eddy Simulation (LES), RANS equations, and Hybrid RANS-LES methods (HRLM) are the three main CFD methodologies [103][104]. To calculate the Reynolds stresses in the RANS equations, these CFD methodologies use turbulence models. RANS techniques at various levels of complexity are “one-equation Spalart-Allmaras (SA) model, two-equation models such as k - ϵ based models, k - ω models, k - ω _SST model, and the complex Reynolds stress model (RSM)” [105]. The energy containing large eddies are solved using LES methods, but it is only possible to run simulations for sub-grid scale motion. As compared to RANS methods, the results obtained from LES is more accurate although it is very time consuming in terms of computational resources [106].

3.2.2 The Spalart-Allmaras model

To make aerofoil predictions the Boeing company created a one-equation Spalart-Allmaras (SA) closure model, which is also the baseline and low Reynolds number model for the first application of the detached eddy simulation (DES) approach [107][108]. This model resolves the viscosity-affected region at the boundary layer. However, the SA model will not capture flow separation, but it is a very stable model, and it will give better convergence. At low intake wind speed up to 7 m/s, the fully turbulent SA model provided acceptable high performance over the blade span due to very less boundary layer separation. At wind speed starting at 10 m/s the SA model could not recreate the flow separation at stall conditions and a mismatch is observed with “surface pressure distribution and low speed shaft torques”. $\gamma - R_{e_{\theta t}}$ transition model was added to SA model where γ stands for intermittency and $R_{e_{\theta t}}$ is the local transition onset momentum thickness. This newly developed $\gamma - R_{e_{\theta t}} - SA$ model provided the same results for the blade operating in both low and high wind speeds. This model can capture laminar flow separation on the blade surface, making it a better performing model at medium intake wind speeds [109].

3.2.3 The $k-\varepsilon$ model

For a turbulent flow with a high Reynolds number, the $k-\varepsilon$ model is very often used by CFD researchers and industries because it gives better convergence along with low memory consumption. The $k-\varepsilon$ model is a two equation model where, k is kinetic energy and ε (epsilon) is the rate of dissipation [110].

Standard $k-\varepsilon$ model: Two independent transport equation are solved by the $k-\varepsilon$ model to calculate the turbulent length and time scale. In ANSYS FLUENT, the standard $k-\varepsilon$ model is classified under $k-\varepsilon$ model and is one of the most used models to calculate the real time engineering flow physics [111]. Assumptions made when deriving the $k-\varepsilon$ model is that the flow is completely turbulent, and the molecular viscosity effects are neglected. As a result, the standard $k-\varepsilon$ model is only applicable for completely turbulent flows. From the known pros

and cons of the standard k - ε model, its performance has been improved with the RNG k - ε model [112] and the realizable k - ε model [113] implementation.

RNG k - ε model: Using renormalization group theory (statistical approach), the RNG k - ε model was developed. Standard k - ε and RNG k - ε models are alike, but the latter has few additional improvements:

- The ε equation has an extra component which enhances the rapidly strained flows accurately.
- The swirling flow accuracy is improved.
- An analytical equation can be obtained using the RNG theory for turbulent Prandtl numbers, whereas a user-specified (constant value) is considered in standard k - ε model.
- Low Reynolds number effects are considered.

Therefore, compared to the standard k - ε model, the accurate and efficient RNG k - ε model is preferable for great variety of flows [114].

Realizable k - ε model: Two additional equations are implemented in the realizable k - ε model, compared to the standard k - ε model:

- For turbulent viscosity and
- For mean-square vorticity fluctuation

The word realizable refers to the model's ability to satisfy specific mathematical limits on Reynolds stresses that are suitable with turbulent flow physics. These abilities are not feasible neither by the standard k - ε model nor by the RNG k - ε model.

3.2.4 The k - ω SST model

The most popular two equation eddy-viscosity model, the k - ω SST (Shear-Stress-Transport), has been developed by Menter [115][116]. The k - ω SST model is an integration of the standard k - ε and the k - ω models, in the free stream and near the wall, respectively. Hence the combination of these models will modify/adjust the flow separation. For unfavourable pressure difference [115] which leads to flow separation a better estimation of the turbulent shear stress can be accounted by the k - ω SST model. By a given KE the boundary

layer within it can be constrained to turbulent shear stress due to modification of eddy viscosity by the $k-\omega$ SST model. Therefore, an unfavourable pressure difference of a high Reynolds number flow is well captured by the $k-\omega$ SST turbulence model [117].

3.3 Wall Function for a Turbulent Model

The highly complex turbulent flow behaviour near the wall has various zones hence, the idea of y^+ was developed. The dimensionless unit y^+ is a distance which is measured from the wall of the model with respect to viscous lengths [118].

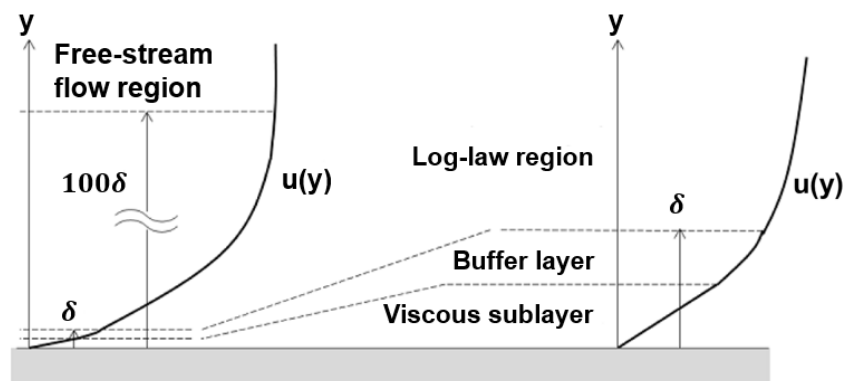


Figure 3.3.1: Regional classification of near wall turbulence flow [119]

From Figure 3.3.1 the four zones near the wall of a turbulent flow are as follows:

1. **At the wall:** Where the fluid velocity is zero
2. **Viscous sublayer:** The distance from the wall to thin layer where the flow velocity is linear and shear stress is dominant which is also known as **laminar sublayer**
3. **Buffer layer:** Above the viscous sublayer turbulence stresses will dominate the viscous force.
4. **Log-law region:** In this zone the flow will be fully turbulent, and the average velocity of the flow can be calculated using logarithmic variant of the distance from the wall.

After the log-law zone the flow reaches to the free-stream zone. δ is distance from the wall to buffer layer which is a thin zone as compared to log-law zone. The log-law region extends to about 100δ from the wall [119].

3.4 Mesh Generation Strategy

The discretisation is one of the most important steps to be able to perform computational studies on any structure/body. The fluid domain is split into several smaller sub-domains which are called elements or cells. Each of these sub-domains are then subjected to governing equations based on the type of study being carried out. Meshing is often considered to be the most crucial part in any numerical study as there are many factors that needs to be considered before generating the final mesh. For instance, the availability of computational resources, reliability, and accuracy of the results which are then evaluated based on the essential grid independence study. As CFD results must be independent of the grid size, the ideal grid size is usually assessed by plotting results obtained with various grid sizes for the individual model. Even though a fine mesh might converge to a solution, there is a limitation for further mesh refinement as this will have an impact on the computational resources being utilised to run the simulation.

To obtain accurate and consistent results, it is important to have a high-quality mesh, which in-turn reduces the computational cost and time by improving the time required to attain convergence [110]. Structured and unstructured mesh are well handled by the current computational programmes. The primary interest of mesh generation is to eliminate the non-orthogonality and skewness of the mesh. To capture the flow physics around the aerofoil it is crucial to implement O-grid or C-grid. However, beyond this region, the remaining part of the domain can be filled by Cartesian blocks [120][121]. Cartesian blocks are the simplest type of structure grid with high quality and are quicker to generate [110].

3.5 Two-Dimensional CFD

3.5.1 Introduction

At the initial stage of the research two-dimensional studies of the aerofoils used in the “DTU 10 MW reference wind turbine” (RWT) [12] were carried out. In this section of the chapter the key findings, and methodology used to validate the 2D simulations are presented.

FFA_W3 has experimental results for 30.1% and 36% thick aerofoils at Reynolds number, $R_e = 1.6 * 10^6$. But in the current study the Reynolds number varies from $R_e = 6 * 10^6$ to $R_e = 12 * 10^6$, therefore the studies were carried out using CFD to determine the aerodynamic characteristics of these aerofoils.

The primary interest of this study is to compare the results of CFD, QBlade with those from the baseline model [12], and to understand the flow physics around the aerofoils. These comparisons will give a better understanding of the percentage of error and the reason behind it.

The steps considered to determine the CFD results are as follows:

1. Create the aerofoil geometry
2. Determine the flow domain
3. Generation of the grid in ICEM-CFD
4. Apply boundary conditions and solver settings
5. Perform flow simulations

3.5.2 Aerofoil Geometry

The “DTU 10 MW RWT” has a relatively light weight rotor which has aerofoils with relative high thickness which increase the structural stiffness. Openly available aerofoils like FFA-W3 series are used. These series of aerofoils are most often selected for the modern large-scale offshore wind turbine designs. The relative thickness of FFA aerofoils varies from 21.1 – 36.0%. In the current studies the aerofoils with relative thickness 24.1%, 30.1%, and 36% were selected as shown in Figure 3.5.1. To have a smooth interpolation between the cylindrical shape at the blade root and the 36% thick

aerofoil, a 48% and 60% thick aerofoils were designed by multiplying the coordinates of the 36% thick aerofoils [12].

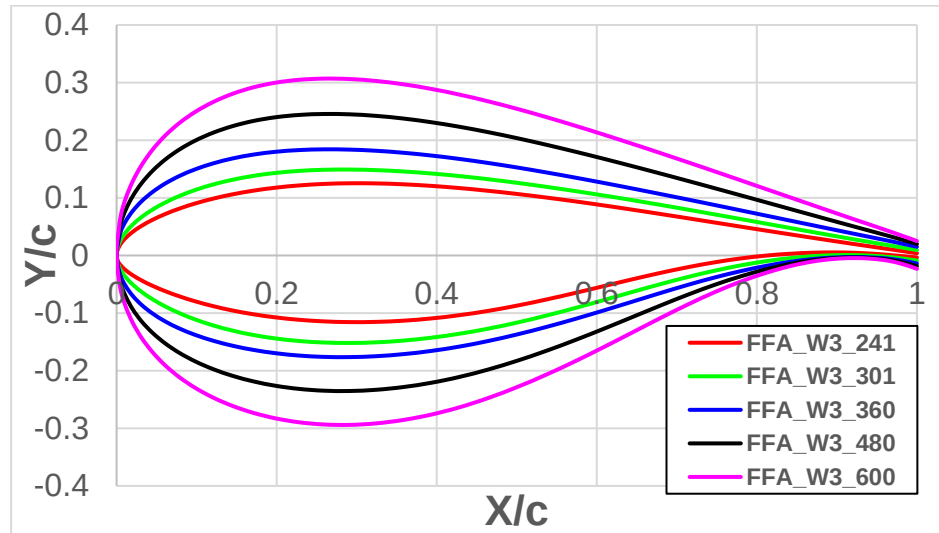


Figure 3.5.1: FFA-W3 series aerofoils used for the 2D studies

3.5.3 Flow Domain Modelling

Around the 2D aerofoils the flow domain was created by extending the boundaries in four directions. A C-shaped pressure-far-field domain was generated as shown in Figure 3.5.2. Various angles of attack can be considered with this domain. To avoid the external flow influence on the aerofoil the downstream side of the domain was placed far away giving sufficient distance to allow the pressure to reach the atmospheric pressure. The distance from the aerofoil to the upstream, above, and below are at 20 chords. The downstream was placed at distance of 40 chord lengths from the aerofoil. The design part of the aerofoils and domain were created in ICEM-CFD.

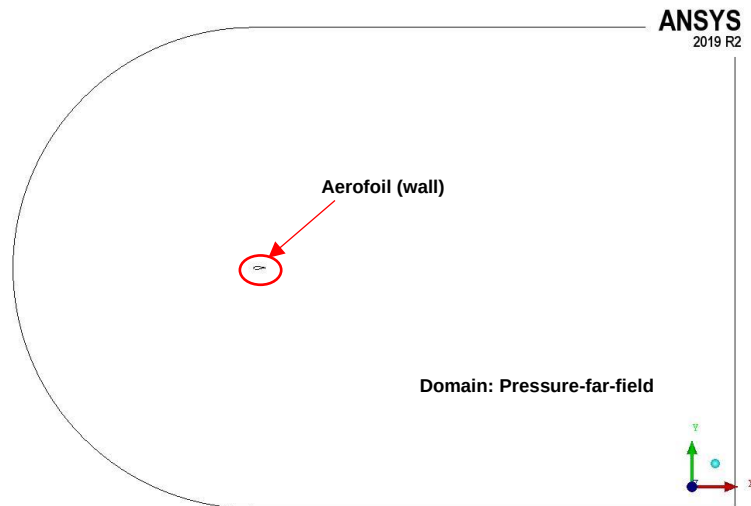


Figure 3.5.2: 2D fluid domain

3.5.4 Grid Generation using ICEM-CFD

A structured Hexa-mesh was generated around the aerofoil using a blocking topology. To obtain the smooth curved mesh around the leading edge of the aerofoil, an O-grid technique was used. Around the aerofoil, a fine mesh was created with a $y^+ = 1$ to achieve a high-quality mesh. The aerofoil and the domain are a single body and the total number of elements generated is 80,540. Mesh quality is essential to capture the flow physics around the blade and to understand the technical reasons behind the flow behaviour. Figure 3.5.3 shows the mesh around the aerofoil and in the domain. The pre-mesh is then converted to an unstructured mesh to generate the output-mesh.

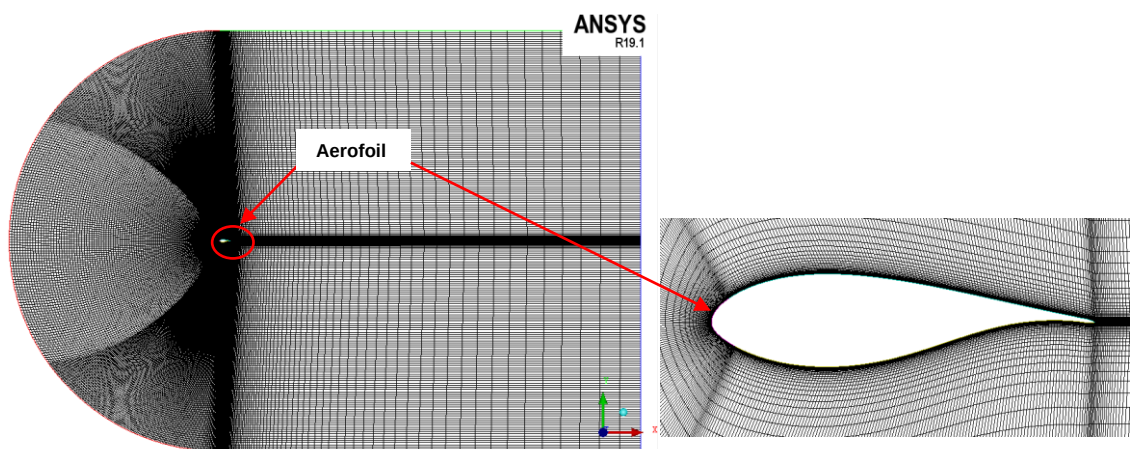


Figure 3.5.3: Mesh generated in the domain and around the aerofoil

3.5.5 Boundary Conditions and Solver Settings

The imposed boundary condition controls the given flow in the fluid domain. Table 3.5.1 shows the FLUENT solver settings. The fluid flow should flow around the aerofoils hence the boundary of the aerofoil is selected as ‘wall’ therefore no flow is penetrating the aerofoil. A steady state, incompressible 2D simulations was carried out for the current case. The domain is set as pressure-far-field because each FFA-W3 series aerofoils are studied at different Reynolds number. Setting the domain as pressure-far-field will allow to set the Mach number (M) which is calculated from the given Reynolds number.

$$R_e = \frac{\rho v_1 C}{\mu} \quad (3.1)$$

From the Reynolds number definition ($R_e = 6 * 10^6$ to $R_e = 12 * 10^6$), the velocity v_1 (m/s) can be calculated. At sea level density is $\rho = 1.225 \text{ kg/m}^3$, the dynamic viscosity is $\mu = 1.78406 * 10^{-5} \text{ kg/(m.s)}$ and the chord length is $C = 1 \text{ m}$.

From the calculated velocity v_1 and known speed of sound c (m/s), the Mach number can be calculated.

$$M = v_1 / c \quad (3.2)$$

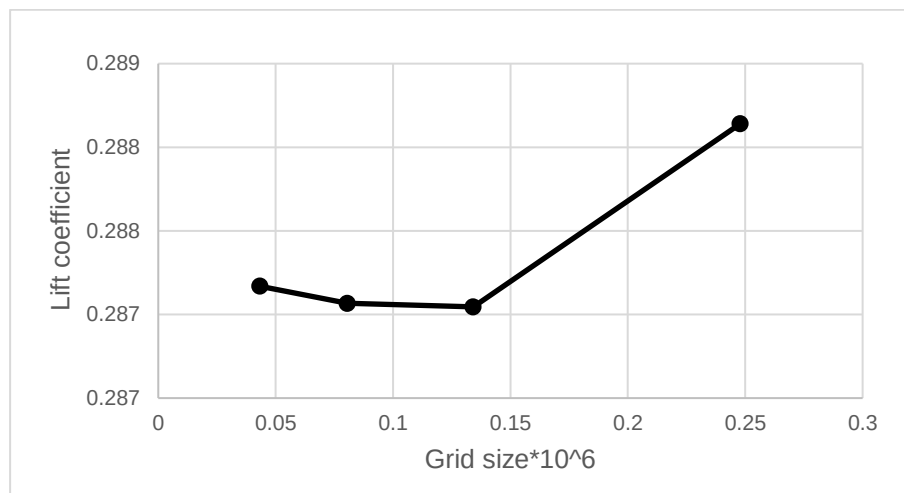
Table 3.5.1: FLUENT solver settings

Solver		Pressured based
Velocity formulation		Absolute
Viscous model		Standard $k-\epsilon$ model
Material		Fluid (air)
Boundary condition		Pressure-far-field
Pressure -velocity coupling		Coupled
Discretization	Pressure	Second order
	Density	Second order upwind
	Momentum	
	Turbulence kinetic energy	First order upwind
	Turbulence dissipation rate	

3.5.6 Grid Sensitivity Study

The grid sensitivity study was carried out to make sure that the results were independent of the size of the grid. Grid/mesh used to discretize the space around the aerofoil will ensure the accuracy of the CFD results. Poorly constructed meshes or an insufficient mesh resolution will lead to grid dependent results. Henceforth, it is important to find the right size of mesh to obtain results that do not depend on the grid density. The grid sensitivity study is therefore a crucial step in a CFD study.

The mesh near the aerofoil is very fine and it becomes gradually coarser away from the aerofoil. This method is also cost-effective. Four different grid sizes were created, made in increments of 1.8 times, starting with 43,360 elements up to 247,844 elements. The lift coefficient and moment coefficient results for all four grid sizes were plotted to choose the optimum one. From Figure 3.5.4, a mesh size between, 80,540 and 134,184 can be selected for further studies because the results of these grid sizes are close to each other. For further studies, the grid size of 80,540 was however selected to reduce the computational time.



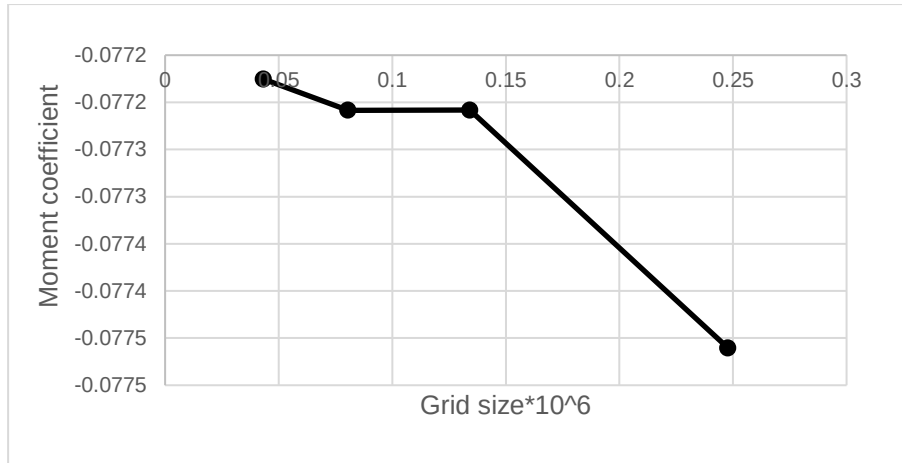


Figure 3.5.4: Comparison of lift coefficient and moment coefficient variation with grid size of FFA-W3-241 aerofoil

Figure 3.5.5 and Figure 3.5.6 shows the pressure contour and pressure coefficient plots of FFA-W3-241 thick aerofoil for 4 different grid size. The plots show negligible difference between each grid size.

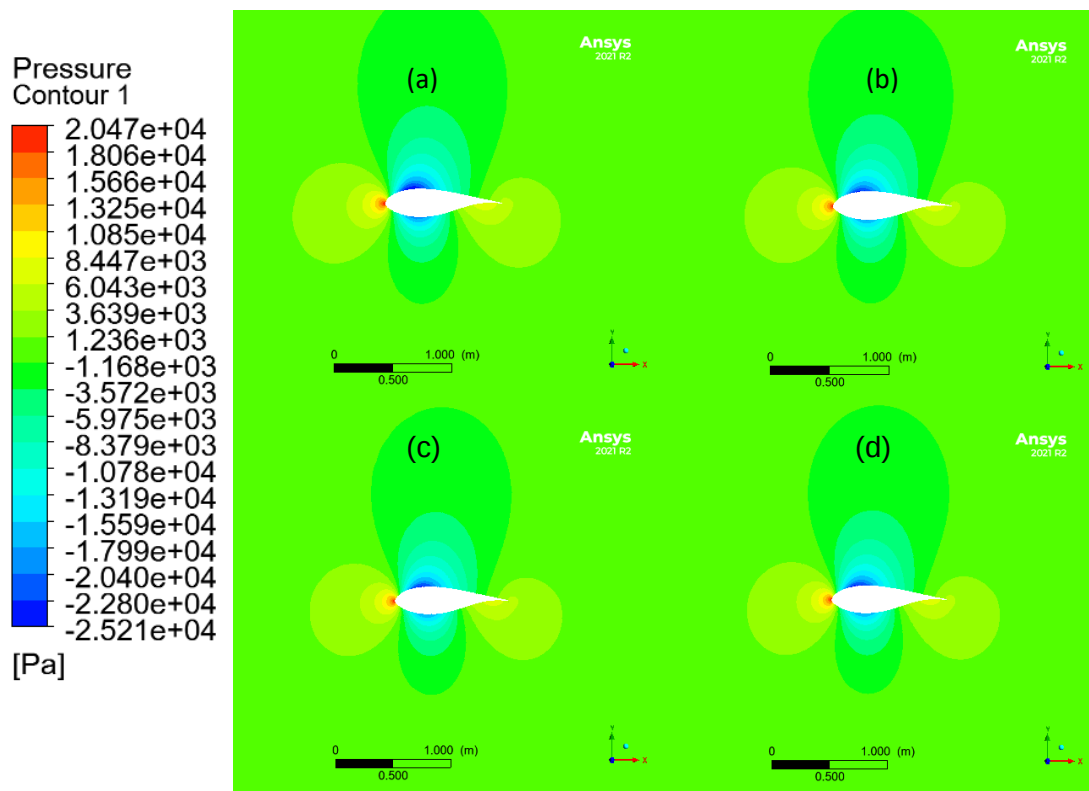


Figure 3.5.5: Pressure contour for grid size (a) 43,360 (b) 80,540 (c) 134,184 and (d) 247,844

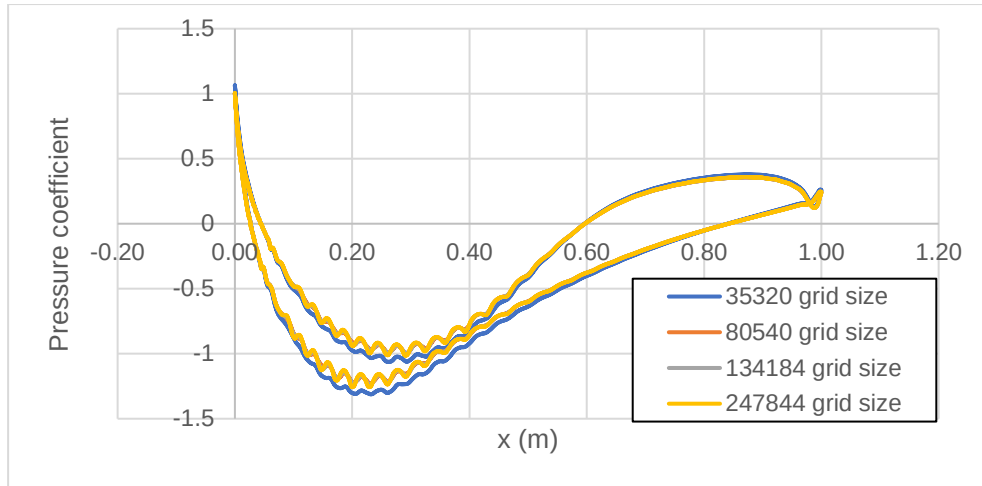


Figure 3.5.6: Pressure coefficient for different grid size

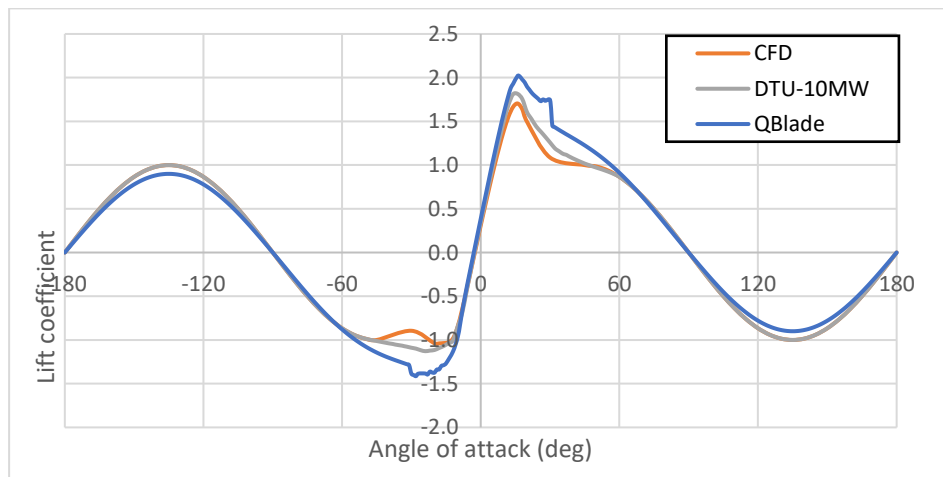
3.5.7 Comparison and Discussion

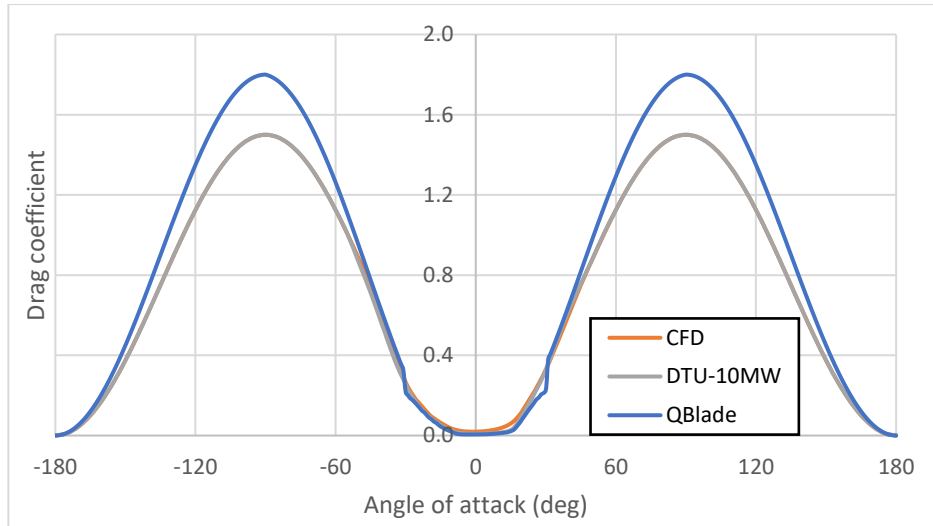
Lift coefficient vs angle of attack plots of various FFA-W3 series are plotted in Figure 3.5.7 at different Reynolds numbers as shown in Table 3.5.2.

Table 3.5.2: Reynolds numbers used for respective aerofoils

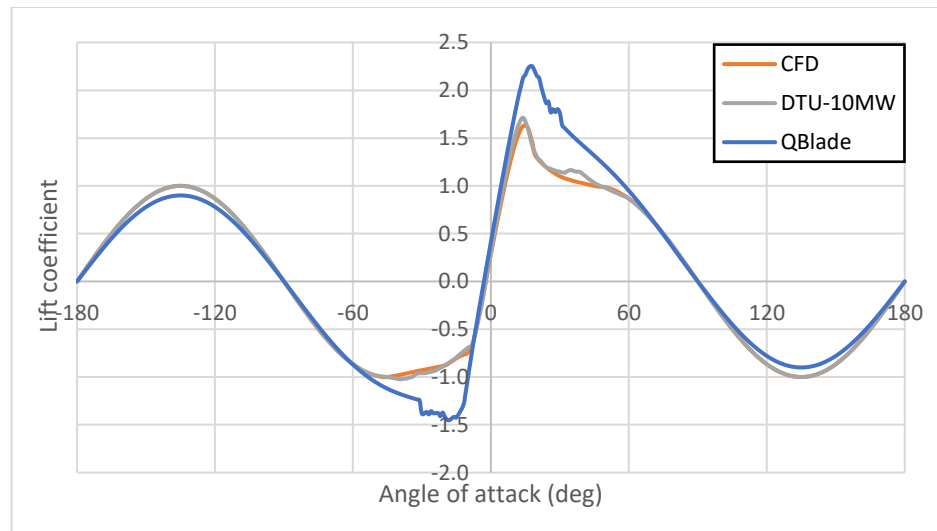
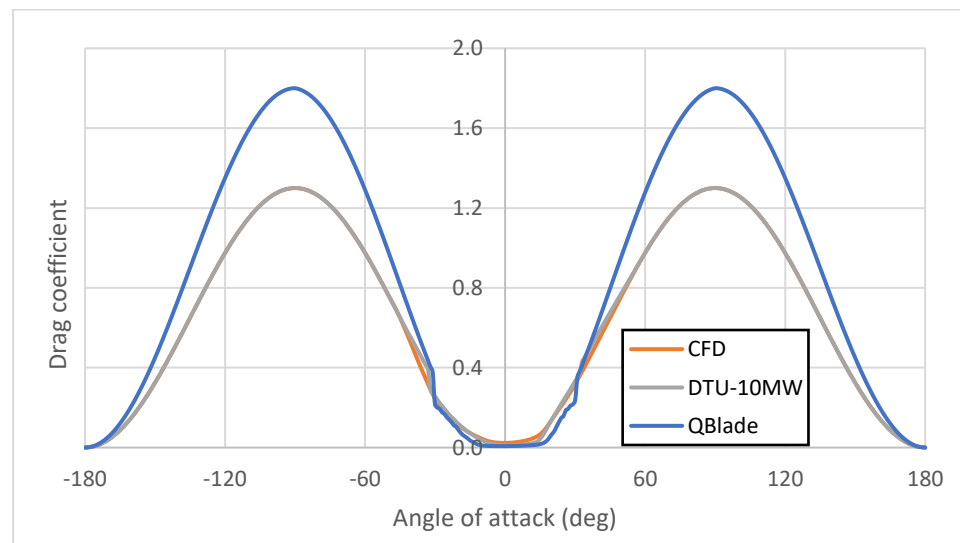
Aerofoils	Reynolds number	Aerofoils	Reynolds number
FFA-W3-241	$12 * 10^6$	FFA-W3-480	$10 * 10^6$
FFA-W3-301	$10 * 10^6$	FFA-W3-600	$6 * 10^6$
FFA-W3-360	$10 * 10^6$		

FFA-W3-241- $R_e = 12 * 10^6$

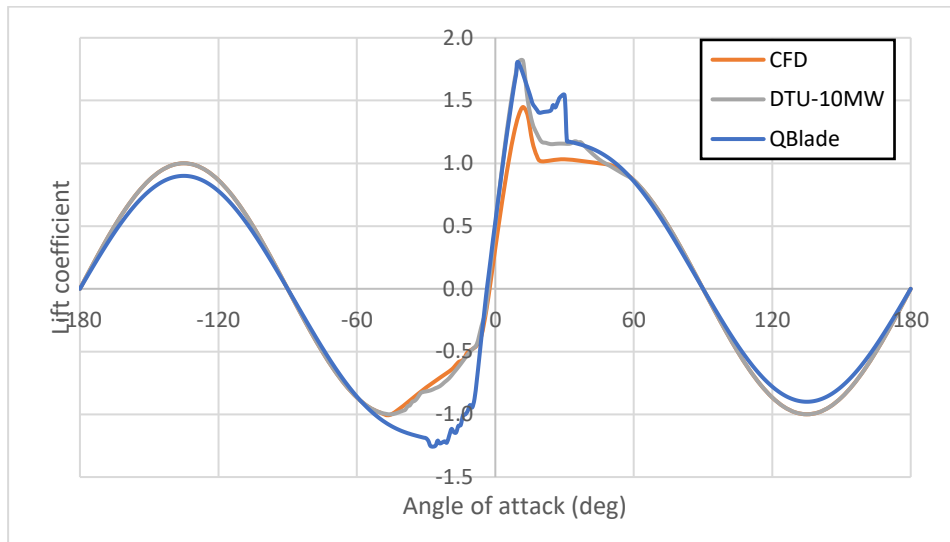
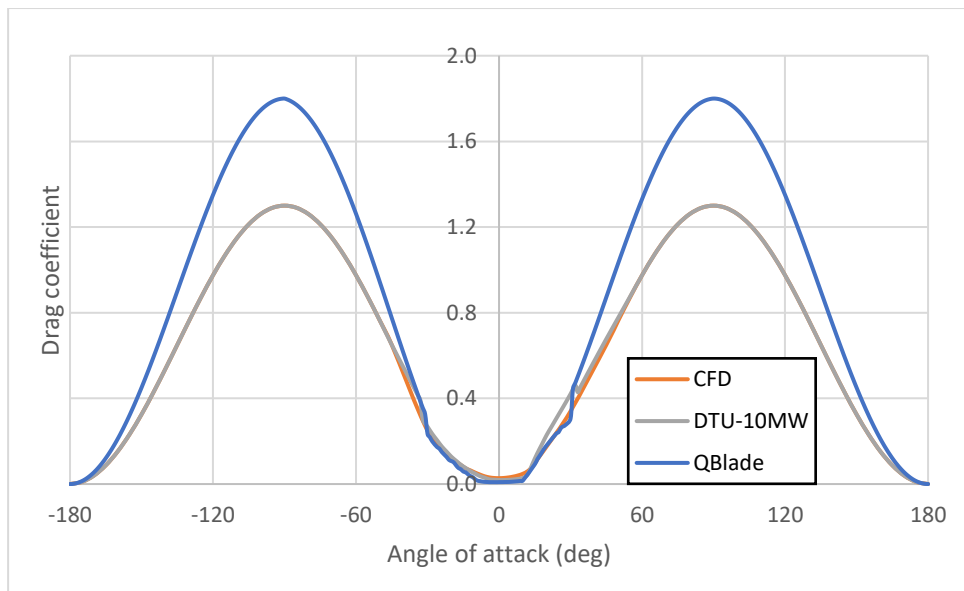




$$\text{FFA-W3-301-}R_e = 10 * 10^6$$



FFA-W3-360- $R_e = 10 * 10^6$



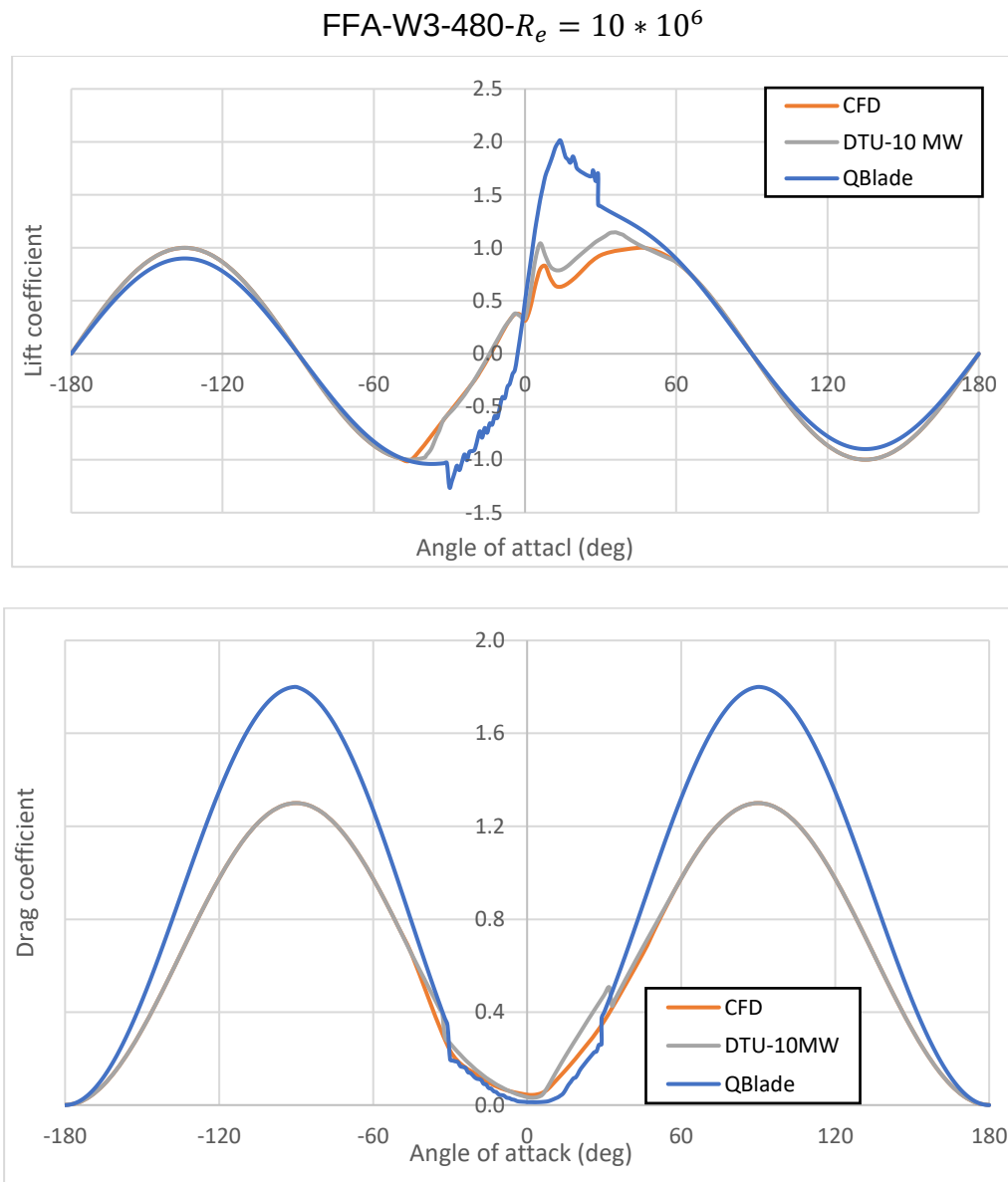


Figure 3.5.7: Lift coefficient vs angle of attack plots for various FFA-W3 series

QBlade results are based on inviscid flow, hence does not match well with the CFD and DTU results. Plots of FFA-W3-360 and FFA-W3-480, there are noticeable differences between the three different software results. This difference is due to the gurney flap which was installed in “DTU 10 MW reference wind turbine [12]” at 36.0%, 48.0% and 60.0% thick aerofoils. The gurney flap will reduce the flow separation at the trailing edge and will compensate the losses due to high relative thickness. In the current study the gurney flap was eliminated to reduce the complexity of the 3D model and to achieve the required power with a simple aerodynamic design. This simplification will also help in cost effective manufacturing and structural stability. From these aerofoils a 3D blade geometry was designed for baseline

study and the minimum C_D/C_L were selected to calculate the chord length and twist angle.

3.6 Three-Dimensional CFD

This section of the thesis involves parametric studies of the baseline wind turbine blade geometry and its performance characteristics. “DTU 10 MW Reference Wind Turbine (RWT)” [12] was selected as the baseline wind turbine, the required results were collected from DTU. The baseline blade and the flow domains were designed in SolidWorks. Then the geometry was exported to ICEM CFD. The required boundary conditions were then implemented, the 3D flow analysis was generated and results post-processed in ANSYS FLUENT. The obtained results were compared with the “DTU 10 MW RWT” to validate the CFD simulation procedure. These studies were simultaneously performed in QBlade, to compare results of CFD, DTU and QBlade. After the numerical modelling, the baseline model was redesigned to obtain the 20 MW wind turbine blade. To do this, the QBlade software was used as a basic design point and to check the power generation from the new designs. This process will provide results in a short time, with reduced computational cost. However, it is worth noting that the results from QBlade cannot be reliable because the code provides a non-viscous solution. Current research proves that CFD simulations are important over QBlade for large scale offshore wind turbines. However, before the comparison of these results, the baseline model grid independence study was carried out to make the solution to be independent of the grid size by running the CFD simulation. Aerodynamics parameters of wind turbine blade like length, chord, twist, RPM, and the pitch were varied to generate a new design model. The geometric models were created for length optimization and chord-twist optimization and CFD simulation were carried out for all the design variants (including RPM and pitch). For each design variant, the interpretations of the solutions are discussed.

3.6.1 DTU 10 MW RWT Design Specification

The “DTU 10 MW RWT” was designed for offshore location and is a three bladed upwind wind turbine. While designing this wind turbine the offshore wind climate was considered to avoid any major issues. This 10 MW wind turbine is an upscaled model of “NREL 5 MW RWT”, but several parameters have been modified, including wind climate, hub height, and the inclusion of prebend blades to ensure tower clearance. Table 3.6.1 shows the design specification of “DTU 10 MW RWT”.

Table 3.6.1: Baseline wind turbine blade design specification [12]

Parameter	DTU 10 MW RWT
Rotor orientation	Clockwise rotation - Upwind
Control	Variable speed
	Collective pitch
Cut in Wind Speed (m/s)	4
Cut out Wind Speed (m/s)	25
Rated wind speed (m/s)	11.4
Rated power (MW)	10
Number of blades	3
Rotor diameter (m)	178.3
Hub diameter (m)	5.6
Rotor speed (rpm)	6 to 9
Maximum tip speed (m/s)	90

3.6.2 Wind Turbine Blade Geometry

The baseline wind turbine blade was selected from the open source [12]. Figure 3.6.1 and Table 3.6.2 give the detailed geometrical information.

Note: The twist of the blade varies from tip to root, the defined twist is positive nose up.

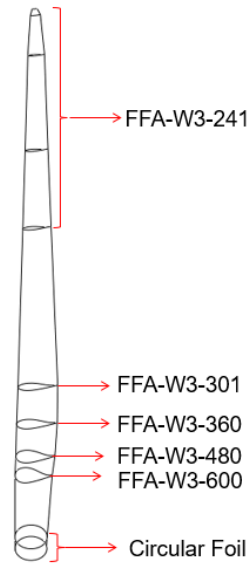


Figure 3.6.1: Baseline wind turbine blade geometry

Table 3.6.2: Baseline wind turbine blade geometry details [12]

Radial Position [m]	Chord Length [m]	Twist [degree]	Thread Axis in [% chord]	Aerofoil Name
0.00	5.38	0.00	0.50	Circular foil
2.80	5.38	-14.50	0.50	
16.87	5.87	-12.55	0.44	FFA-W3-600
20.88	6.11	-9.96	0.40	FFA-W3-480
27.49	6.18	-7.42	0.37	FFA-W3-360
34.66	5.89	-5.92	0.35	FFA-W3-301
61.05	3.79	-0.72	0.35	FFA-W3-241
71.97	2.91	1.11	0.35	
83.86	2.02	2.80	0.35	
88.45	1.18	3.36	0.35	
89.17	0.60	3.43	0.35	

From the existing coordinates of FFA-W3 series aerofoils, the required chord length and the calculated twist were added, keeping the thread axis between 0.5 – 0.35% of the chord using the QBlade software. The total diameter of the hub is 5.6 m and the blade is placed in such a way that the root of the blade (circular foil) completely immerses into the hub as shown in Figure 3.6.2. For the simplification of the study, a single blade was selected.

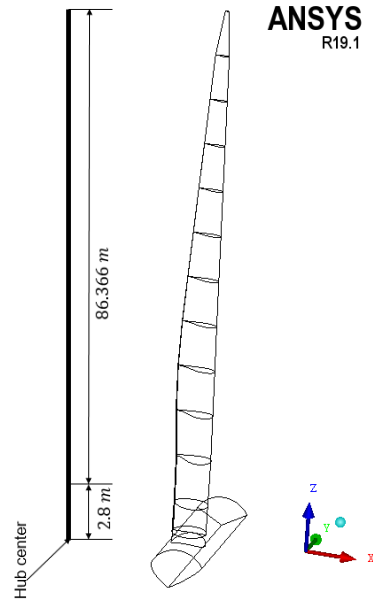


Figure 3.6.2: Blade geometry for CFD simulation

3.6.3 Flow Domain Modelling and Sensitivity Study

Around the wind turbine blade the flow domain was created by extending the domain upstream and downstream to a fixed distance using Workbench design modular. Considering a steady flow around the blades that are placed symmetrically at an angle of 120 degree , periodic boundary condition can be imposed in the other two direction. Hence a single blade simulation can be carried out for the research work. When using a moving reference frame with a wind turbine blade geometry in CFD, it is recommended to connect the fluid interface, which is an effective and accurate method. The blade was placed inside two domains (main and sub-domain) in-order to create the moving reference frame mode. This reference frame will simulate the flow around the blade.

Two different shaped domains were created to understand the domain sensitivity and to determine the domain size and flow influence. Regardless of the turbulence model, the size of the domain must be large enough to minimize the interference caused by boundaries. This interference will affect the aerodynamic forces of the wind turbine blade. The distance between the inlet and the blade must be long enough to develop the wake flow. It is important to keep in mind that any unnecessary increase of the domain size will adversely

affect the computational cost. Henceforth the domain size (computational cost) and accuracy of the results (aerodynamic force) are contradictory topics, and a wise decision must be taken. One of the solutions to this is to consider periodic boundary conditions.

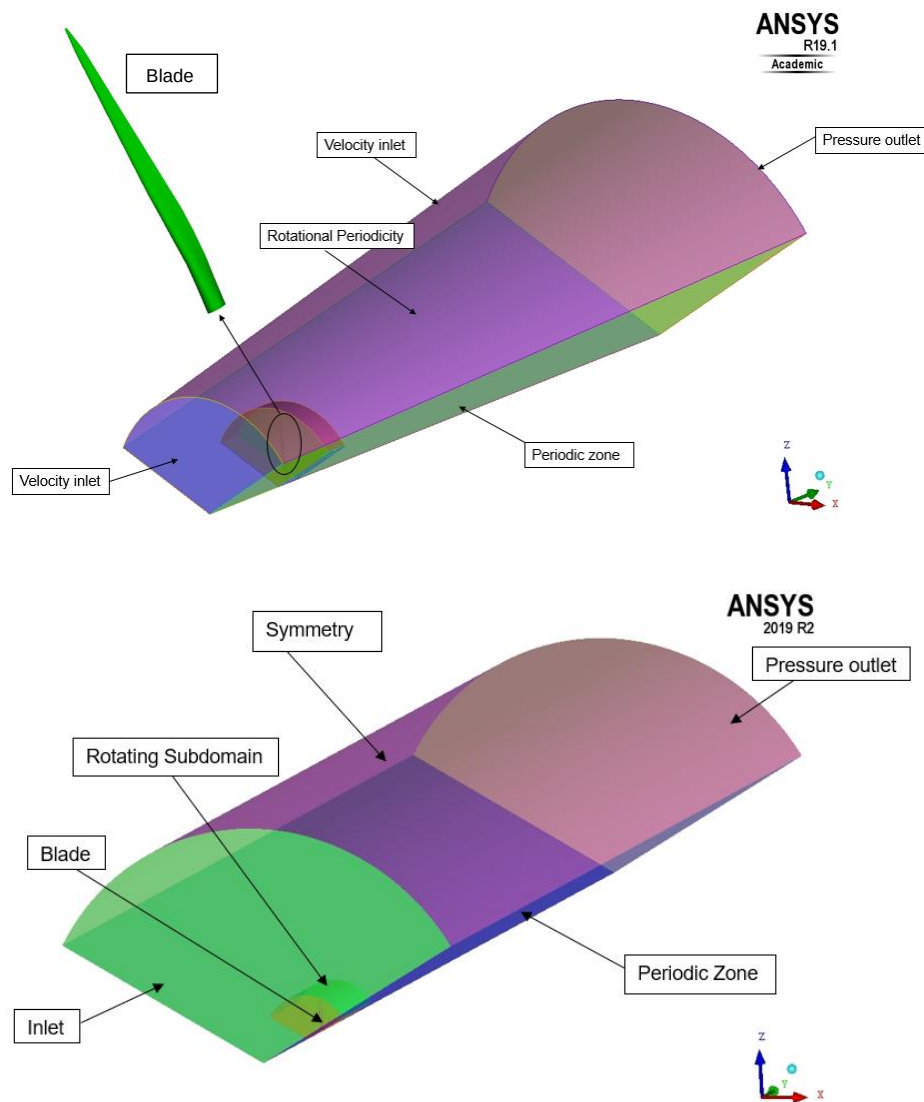


Figure 3.6.3: Conical and cylindrical shaped domain

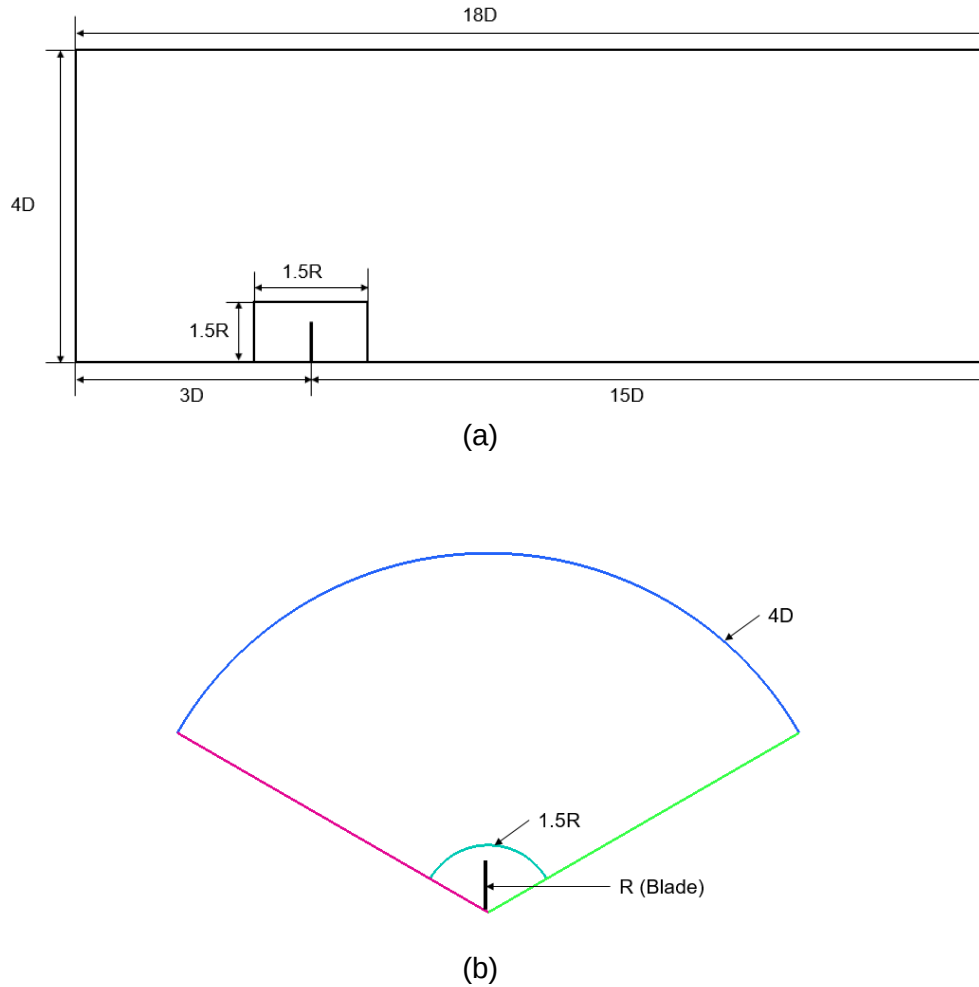


Figure 3.6.4: Dimensions of CFD cylindrical domain - (a) Side view - (b) front view, D -diameter, and R -radius of the blade

A detailed study of the domain size and shape was carried out. A conical and cylindrical domain shape were studied, as shown in Figure 3.6.3. Figure 3.6.4 shows the side and front view of the domain with dimensions. The cylindrical domain was selected based on the results obtained and the flow physics near the outlet of the domain. Table 3.6.3 gives the details of the dimensions used to determine the effective size of the domain and the turbulence model to be used to obtain the cost-effective accurate results (Appendix 7.1).

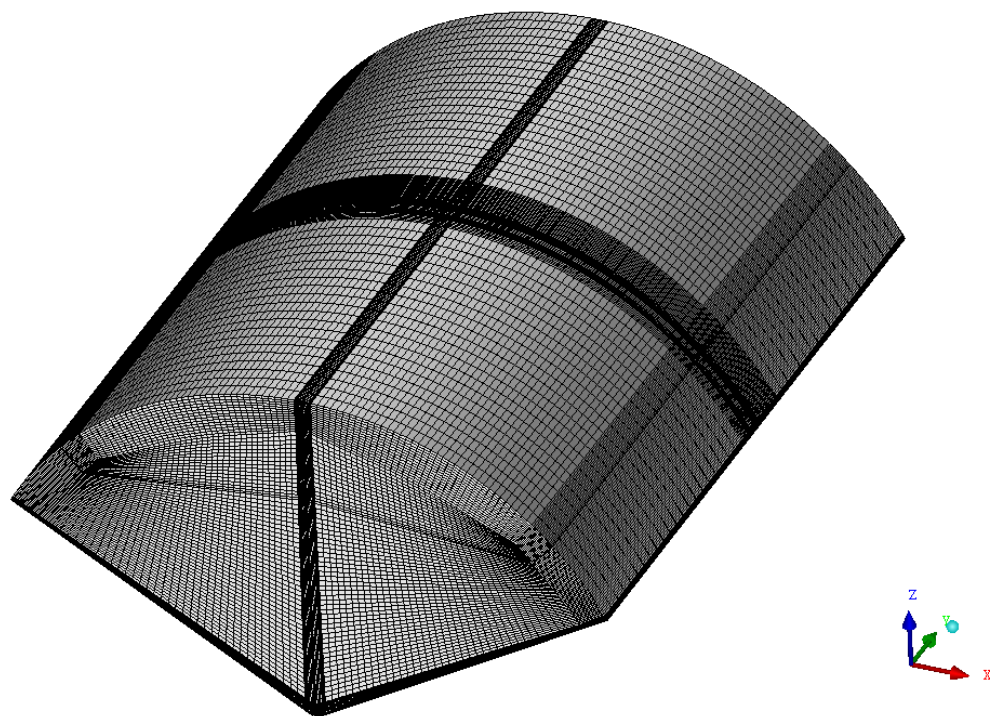
Table 3.6.3: Boundary condition interaction and placement sensitivity

Boundary Conditions	Cases	Selected
Domain Shape	Conical, and Cylindrical	Cylindrical
No' of blade	1 blade, 3 blade	1 blade
Inlet Boundary (velocity inlet)	3D, and 6D upstream	3D upstream
Vertical surface of computational domain (Symmetry)	4D, 7.5D, and 12D	4D
Outlet Boundary (pressure outlet)	7D, 10D, 15D 20D downstream	15D downstream
Subdomain	1.5R, and 2.5R	1.5R
Time	Steady, and Transient	Steady
Models	Laminar, Spalart-Allmaras, k-epsilon (standard and realizable), k-omega SST, transition k-kl-omega, and DES	K-omega SST
Rotating domain	Outer and subdomain, subdomain only	Subdomain only

3.6.4 Grid Generation

Both the outer and the subdomain were generated in Workbench design modular and the .iges files were imported into the ICEM CFD meshing software. These two domains were meshed separately and saved as .msh files. Meanwhile the blade was divided into two parts as bottom and top, with chord lengths of 0.6 m and 5.38 m at the tip of the blade and at the root, respectively. The total element size for the model is 8.3 million which are divided as follows; outer domain – 3.8 million, inner domain – 1.5 million, bottom blade – 2.3 million and top blade – 0.5 million. Different boundary conditions, velocity inlet,

pressure outlet, symmetry (vertical surface of the computational domain), stationary wall (blade and hub), periodic plane, outer domain and sub domain as interface were set. Due to numerical diffusion, computational costs, and time taken to generate the grid; it is crucial to select the type of grid to be used. A grid can either be structured or unstructured. Generating a structured grid on a complex geometry is more time-consuming than an unstructured grid. However, on a blade, a structural grid can be generated, hence avoiding the false numerical diffusion which depends on the grid alignments. In the current research, to have a Hexa mesh, a blocking topology was performed on the outer and subdomain using an O-grid technique which generates a circular mesh, as shown in Figure 3.6.5.



(a)

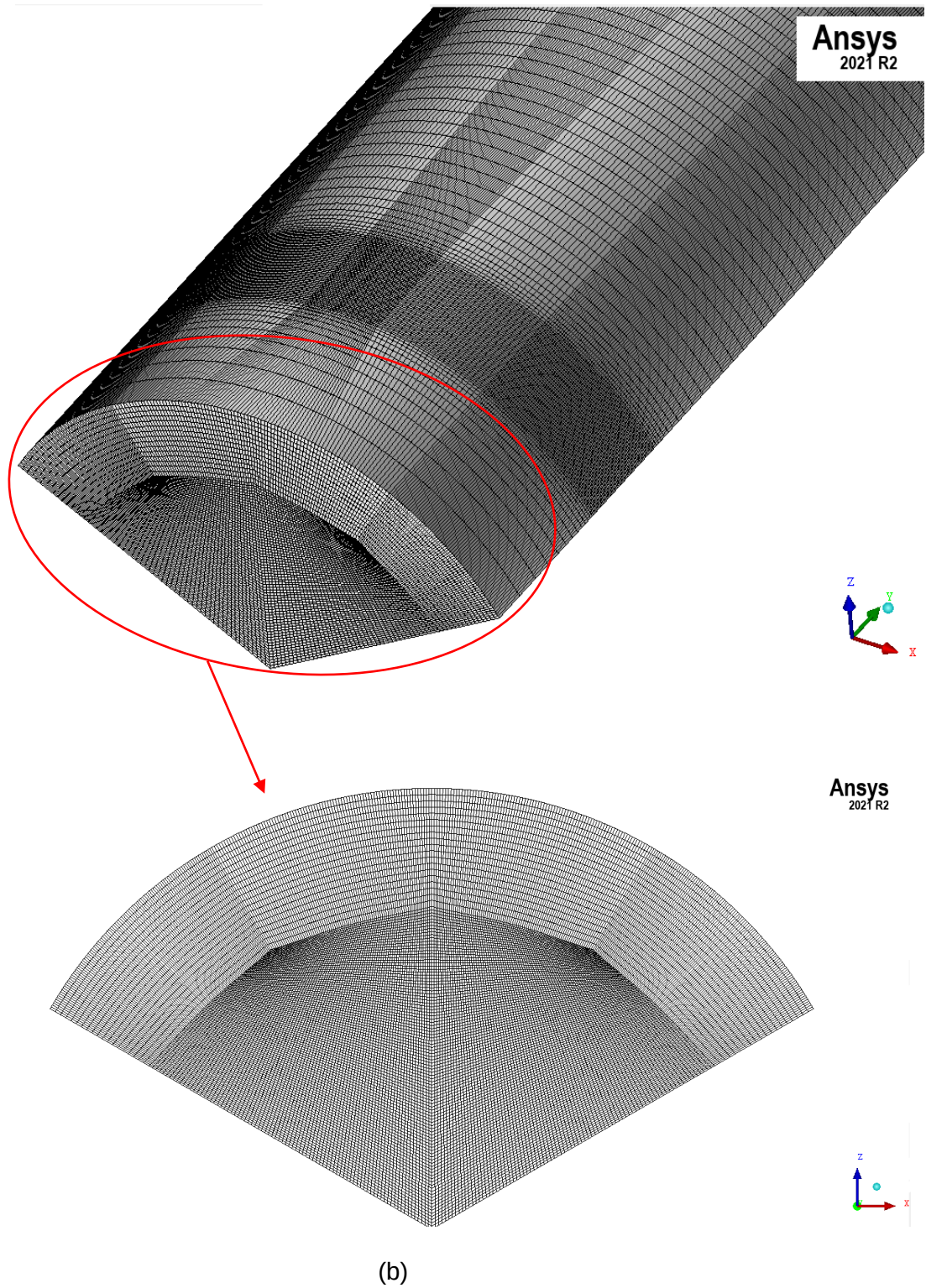


Figure 3.6.5: Mesh on (a) subdomain and (b) outer cylindrical flow domain

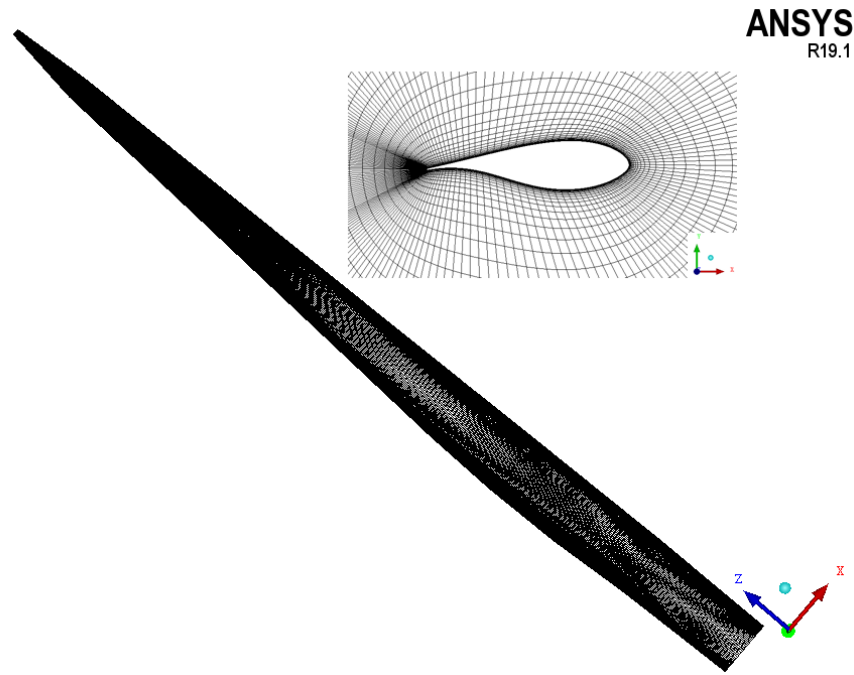


Figure 3.6.6: Blade mesh and mesh around blade aerofoil at an intermediate section (80% span)

A Hexahedral grid can be easily re-meshed, which provides projection-based grid generation environments. This methodology will project the edges, vertices and faces to the nearest surface of the new CAD geometry and by reassociating the projected mesh with the new geometry, a mesh can be generated. Later this premesh is converted into a single unstructured mesh, which will merge all the common nodes on the block interface. Hence a hexahedral grid was created around the blade as shown in Figure 3.6.6.

3.6.5 Solver Setting and Boundary Condition

ANSYS FLUENT is a CFD tool designed specifically for fluid flow simulation and heat transfer. This software also includes wide range of modelling applications for academics and industries like multiphase flows, turbulence flow, combustion, and reaction flows. Variety of simulations like compressible, incompressible steady, and transient cases can be studied. ANSYS FLUENT simulations are fast, accurate and cost effective, hence a better decision can be made regarding the design. The FLUENT settings are classified into four major groups.

1. Model settings
2. Material properties
3. Boundary conditions
4. Solver controls

Model settings

The model settings help to define the flow characteristics, such as steady or transient mode, number of equations (laminar or turbulent), compressible or incompressible flow, etc. In the current research steady state time independent analysis was carried out. The tip speed is 90 m/s at 9.6 rpm and the blade operates at a high Reynolds number, hence the flow is turbulent. The calculated Mach number from the highest tip speed is less than 0.3 which makes the flow as incompressible flow. Simulations such as direct numerical simulation (DNS), or large eddy simulation (LES) could be used to resolve the current wind turbine blade model, but it is not suitable for most practical applications because of their high computational cost. The velocity in the turbulent flow is unpredictable and is random hence Reynolds-Averaged Navier- Stokes equations are used, which would help to determine the statistical properties of the flow. This commonly used procedure produces a cost-effective computation. The $k\text{-}\omega$ SST model was considered in the present work.

Material properties

Wind turbine blades convert the kinetic energy of the wind into electrical energy; hence, air is considered as the fluid. As discussed, the flow is assumed incompressible therefore the properties of the air were treated as constant. Defined air properties are at sea level are:

$$\text{Density} = 1.225\text{ kg/m}^3$$

$$\text{Viscosity} = 1.7894 * 10^{-5}\text{ kg/m.s}$$

Boundary Conditions

Using a cell zone condition in FLUENT, the frame motion was enabled, and the subdomain and blade fluids were set to be rotational flow in the axis of rotation ($-y$). The rotational velocity was set according to the velocities, whereas in the outer domain the flow was stationary.

The following boundary conditions were considered:

- Inlet: Velocity-inlet boundary condition was given, and the Y-velocity was defined.
- Outlet: Pressure-outlet, and the gauge pressure was zero pascal whereas operating pressure is 101325 pascals.
- Blade and hub: Stationary wall with no slip.
- The vertical surface of computational domain was symmetry.
- The domain bottom was periodic in both inner and outer domain

In the interface setup tool, the faces of the subdomain (rotational) and the inner domain of the outer domain (as shown in Figures 3.6.3 and 3.6.4) are interfaced. All the required faces of the blades were interfaced with a matching interface option. The periodic bottoms of the domains were interfaced with a rotational type with an offset of 120 *degree*.

Solver Controls

The pressure-based solver and the density-based solver are the two types of solvers available in FLUENT. In these two solvers the governing flow equations are discretized by using the finite-volume discretization scheme. The Equation of State in the density-based solver will determine the pressure field. But the pressure-based solver will determine the pressure field by resolving a pressure or pressure correction equation. This pressure field is obtained by adjusting the continuity and momentum equations which must satisfy the continuity equation by correcting the pressure equation of the velocity field. The governing equations of both algorithms are non-linear and coupled. Energy, continuity, and momentum equations are solved by the density-based solver, this is not the case for the pressure-based approach. In the present simulation,

the pressure-based solver was used to calculate the pressure field since an incompressible flow is considered. To derive the face values of the scalar variables, the first-order upwind discretization approach was used to discretize the convection components in the governing conservation equation. Pressure and velocity are stored at cell centres in FLUENT's collocated approach. The STANDARD pressure interpolation technique was used to acquire the pressure values at the faces. The STANDARD scheme in FLUENT uses momentum equation coefficients to interpolate the pressure values at the faces. The SIMPLE approach, which is FLUENT's default methodology and is an iterative solution method, was used to obtain the pressure adjustment equation. Table 3.6.4 shows the solver parameter settings used in FLUENT.

Table 3.6.4: Solver parameter settings used in FLUENT

Solver		Pressured based
Velocity formulation		Absolute
Time setting		Steady state
Viscous model		k-omega SST
Material		Fluid (air)
Boundary condition	Inlet	Velocity-inlet
	Outlet	Pressure-outlet
	Blade	Stationary wall – no slip
	Vertical surface	Symmetry
Pressure -velocity coupling		SIMPLE
Discretization	Gradient	Least squares cell based
	Pressure	STANDARD
	Momentum	Second order upwind
	Turbulence kinetic energy	First order upwind
	Turbulence dissipation rate	

Residual values of 10^{-6} were set to achieve the convergence, however these solutions did not reach this criterion. Hence, simulation cases were assumed to be converged when:

- All the residuals reached a reasonable steady state value, below 10^{-4} .
- The solution did not alter with further iterations or all discrete conservation equations (momentum, xyz-velocity, k, omega) has been obeyed in all cells within a certain tolerance.
- Residual plots have been used to monitor the convergence.
- Along with residuals, thrust, torque, C_l , and C_d have been monitored to confirm convergence. Figure 3.6.7 shows the convergence plots.

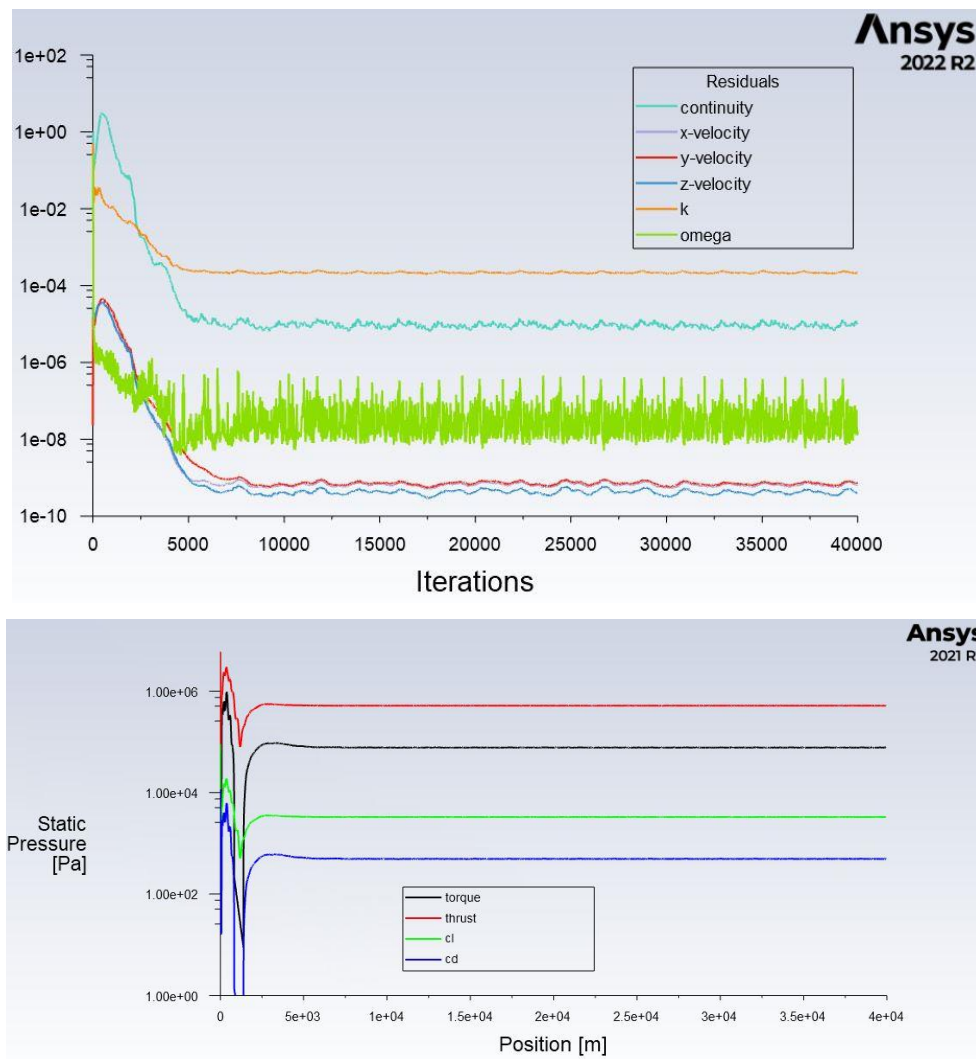


Figure 3.6.7: Convergence plots

3.6.6 Grid Independence Study

Similarly, to the 2D grid sensitivity study discussed in Section 3.5.6, a 3D grid independence study was conducted. A domain sensitivity study was carried

out, as mentioned in Section 3.6.3 to finalise the optimum domain size and boundary conditions. On the finalized domain, grid independence studies were performed.

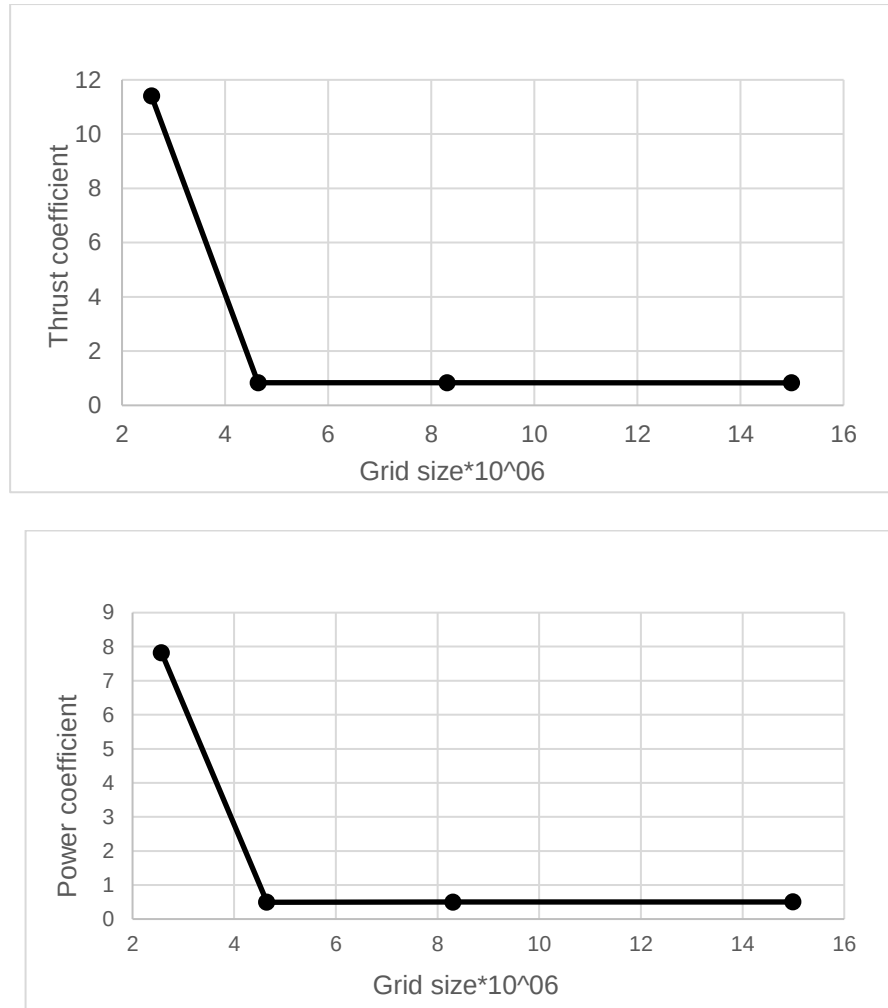


Figure 3.6.8: Thrust and power coefficient results comparison with grid size at a wind speed 11 *m/s*

Four different grid sizes (2.57, 4.64, 8.31, 14.99 million elements) were studied to finalize the number of elements that can be utilized to conduct further studies. For the four grids, a consistent agreement in power coefficient and thrust coefficient were obtained, with the increased grid size. Figure 3.6.8 shows the thrust and power coefficient results against the grid size. The study was carried out at a rated wind speed of 11 *m/s*, with a zero pitch of the blade. From the plot it can be concluded that the minimum grid size required to achieve the solution independent of the grid size is above 4.64 million cells. Henceforth,

in the current research, grid sizes considered were between 4.7 and 8.5 million cells.

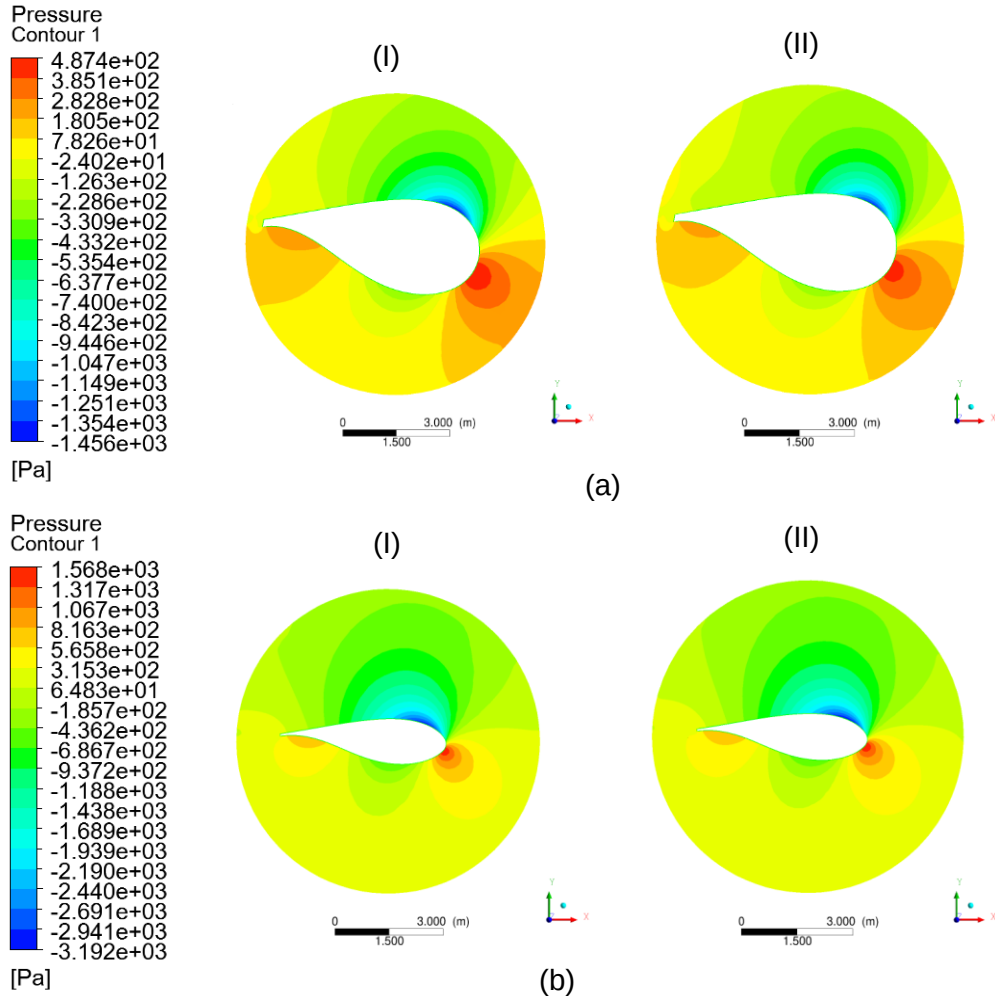
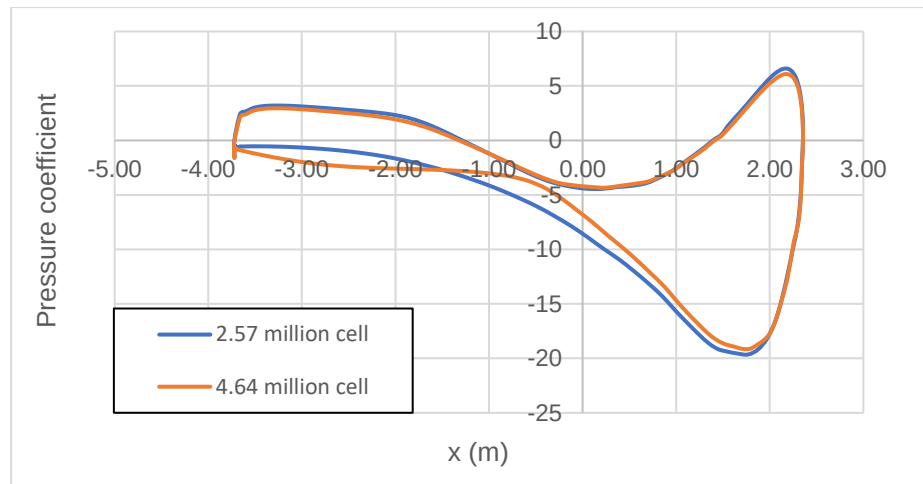
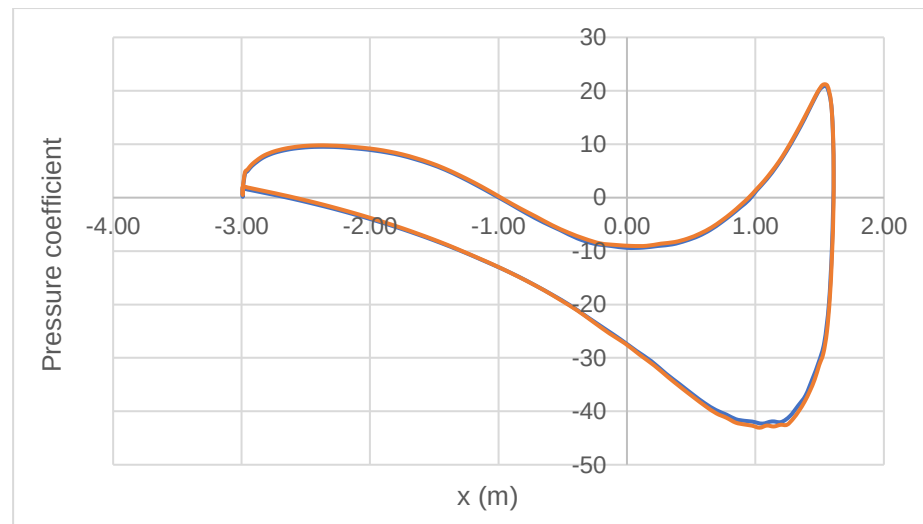


Figure 3.6.9: Comparison of pressure contours for (a) 30% span and (b) 60% span of the blade for grids of (I) 2.57 *million* and (II) 4.64 *million* cells

Figure 3.6.9 and Figure 3.6.10 shows the variation of pressure on the 30% and 60% of the blade span. A fine grid captures better flow physics which in turn results in a higher accuracy of the results. The flow physics are captured for two different grid size of 2.57 million (I) and 4.64 million (II) cells.



(a)



(b)

Figure 3.6.10: Comparison of pressure coefficient for (a) 30% span and (b) 60% span of the blade for grids of (I) 2.57 million and (II) 4.64 million cells

3.6.7 Baseline Wind Turbine Blade Performance

Numerous factors need to be considered to establish limitations in the blade design stage. The “NREL 5 MW Reference Wind Turbine (RWT)” [11] was upscaled and modified by Technical University of Denmark as “DTU 10 MW RWT” [12]. In this model the radius of the blade was 89.166 m and the minimum relative thickness was 24.1%. However, to overcome the structural stiffness issues of a large-scale offshore wind turbine the thickness was increased by

challenging the aerodynamic performances. The “DTU 10 MW RWT” was selected as a baseline model and the required aerodynamic parameters and results were collected from the researchers involved in its design.

The maximum power coefficient was achieved at tip speed ratio (TSR) of 7.5. The TSR value will contribute to the shaft torque and power efficiency, noise, and structural stiffness [12]. The figure 3.6.11 shows the C_p vs TSR of the baseline model at a velocity of 11 m/s.

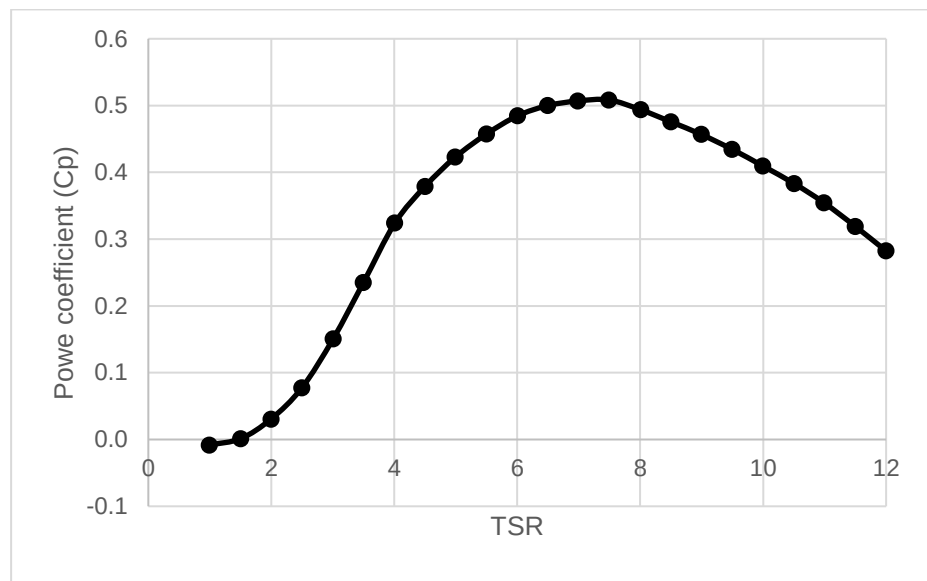


Figure 3.6.11: C_p vs TSR at wind speed of 11 m/s (QBlade)

The baseline model was designed and meshed as explained in previous sections. This meshed geometry was imported in FLUENT, with boundary conditions reported in Table 3.6.4. Design and off-design conditions were studied, and the aerodynamic results were captured. In the current research fully turbulent CFD simulations were conducted. Table 3.6.5 shows the wind speed, pitch, and RPM considered in this study.

Table 3.6.5: Operational parameters for the baseline blade geometry, (pitch-nose down) [12]

Velocity (m/s)	Rotational speed (RPM)	Pitch angle (degree)
5	6	1.966
6	6	0.896
8	6.423	0
9	7.225	0
10	8.029	0
11	8.837	0
12	9.6	4.502
16	9.6	12.499
20	9.6	17.618
25	9.6	22.975

To achieve accurate results, a pitch sensitivity study was carried for the pitched blade velocities. The pitch was reduced between 1 – 1.5 *degree* accordingly to obtain the required results in the CFD study. Table 3.6.6 shows the studies carried out on the pitched blade. An optimized pitch was then selected, and the results were compared. The pitch of the blade will be different due the domain sensitivity and the boundary condition.

In QBlade, the same pitch as mentioned in Table 3.6.5 was used to get the results, and no corrections were applied. For the CFD study, however, a pitch correction was applied to minimise the percentage of error.

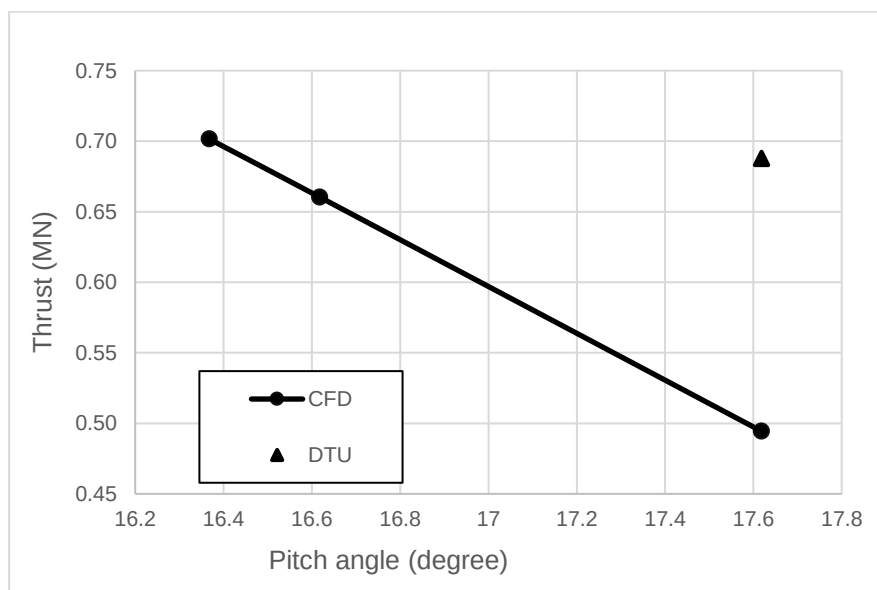
Table 3.6.6: Optimized pitch in current study (CFD)

Velocities (m/s)	Reduction in Pitch angle (degree)	Optimized Reduction in Pitch angle (degree)	Original Pitch angle (degree)	Optimized Pitch angle (degree)
5	1, 1.25	1	1.966	0.966
12	1	1	4.502	3.502
16	1	1	12.499	11.499
20	1, 1.25	1.25	17.618	16.368
25	1, 1.25, 1.35, 1.5	1.35	22.975	21.625

Only velocities of 20 m/s and 25 m/s in pitch sensitivity studies are shown in detail to give a clear picture of thrust and power variation. However, velocities of 5, 12, and 16 m/s (Appendix 7.2) are studied as well, and the optimized results are considered to plot and compare the results for each off-design conditions.

The blade angle is determined from the collective measure of angle of pitch, twist, and angle of attack, this is also known as the angle of relative wind. The twist of the blade is fixed, where the C_D/C_L is minimal so that C_D is almost equal to zero. This allows to achieve a maximum power output, according to Betz' limit. Hence the angle of attack is also fixed because the selected C_D/C_L is at a particular angle of attack. Therefore, the only angle that can be varied to reduce the angle of relative wind, is the angle of pitch. This will allow and distribute the inlet flow parallel to the blade surface, reducing the stall effects and flow separations.

For the 20 m/s velocity, a detailed performance study was carried out. Figure 3.6.12 and Table 3.6.7 show the variation of thrust and power with respect to the pitch. Thrust and power increase as the pitch is reduced, this is due the reduction in stall. As reported in Table 3.6.6, three different pitch angles were studied for 20 m/s (17.618, 16.618, 16.368 degree).



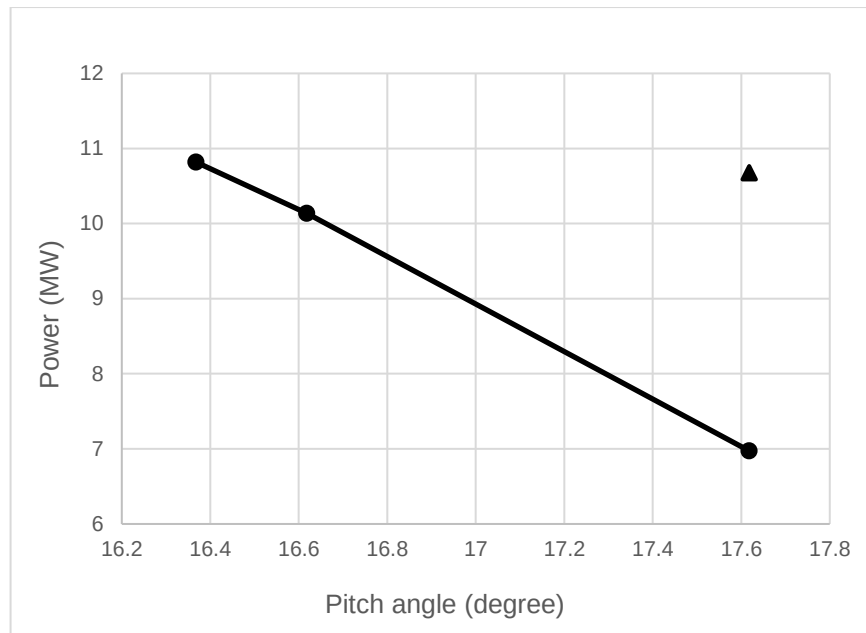


Figure 3.6.12: Thrust and power variation with different pitch angle at velocity 20 m/s

A pitch angle of 16.368 degree was selected according to the minimum percentage of error between the current CFD results and the results of “DTU 10 MW RWT”. The percentage of error was 2% for the thrust and was 1.31% for the power.

Table 3.6.7: Percentage of error for thrust and power at velocity 20 m/s

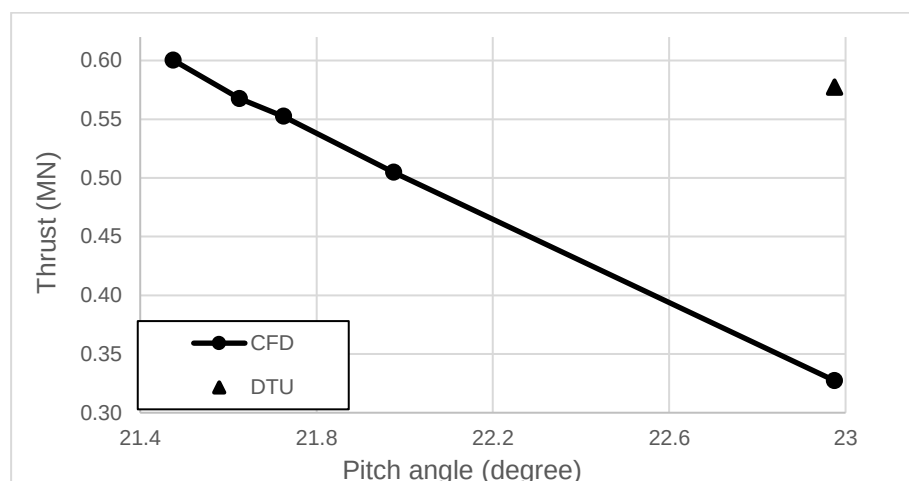
	Pitch sensitivity study		DTU 10 MW RWT		% of error	
Pitch (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
17.618	0.494	6.973	0.6877	10.6785	-28.10	-34.70
16.618	0.660	10.137			-3.98	-5.07
16.368	0.701	10.818			2.00	1.31

For the 25 m/s velocity, five different pitch angles were studied, and the given pitch angle was 22.975 degree . The angle of pitch was reduced from 1 degree to 1.5 degree as shown in Table 3.6.6. The required thrust and power were achieved with a 1.35 degree reduction i.e., 21.625 degree . The optimum pitch angle was selected according to the minimum percentage of error. Thrust

and power increased as the pitch angle was decreased, as shown in Figure 3.6.13 and Table 3.6.8.

The pressure difference on the blade surface produces the thrust, which is an integrated normal force on the blade surface. The required thrust was achieved at a pitch angle of 21.625 *degree* with a percentage of error of 1.7%. Further reduction in the pitch angle beyond 21.625 *degree* will increase the thrust. In an upwind wind turbine, the produced thrust will push the blade towards the tower. If the thrust is increased, then the blade will collide with the tower which will damage both blade and tower. To avoid these issues, the thrust must be accurate.

Similarly, the tangential force will generate the torque from which power is calculated. The accurate power was generated at a pitch angle of 21.625 *degree* with a percentage of error of 0.68%. At a higher velocity, the blade is pitched towards feathering, so that the angle of attack of the blade is influenced. This will reduce the lift of the blade and will control the power. A further increase in pitch angle will stop the wind turbine blade from producing lift and the turbine will stop rotating. Or increasing the angle of attack of the blade to a critical value causing stall intentionally, which is known as active stall above designed wind speed. Therefore, increasing the drag will slow down the blade rotation.



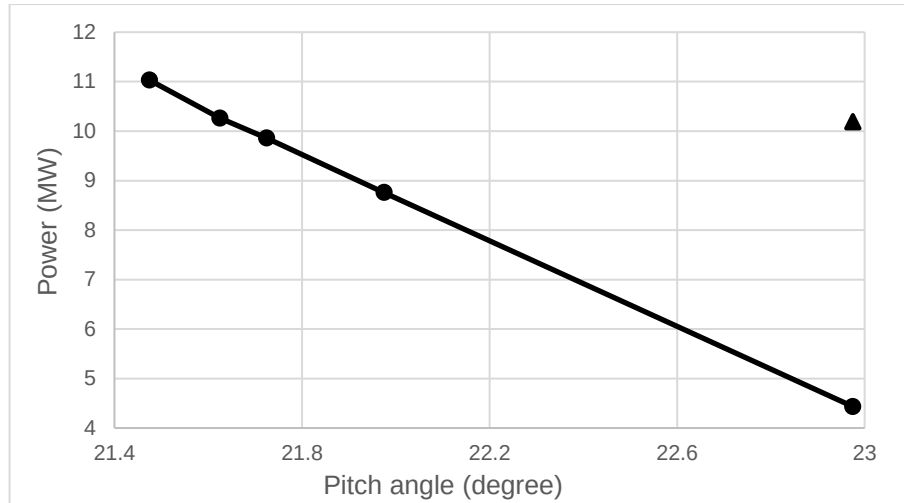


Figure 3.6.13: Thrust and power variation with different pitch angle at velocity 25 m/s

Table 3.6.8: Percentage of error for thrust and power at velocity 25 m/s

	Pitch sensitivity study		DTU 10 MW RWT		Percentage of error (%)	
Pitch (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
22.975	0.327	4.434	0.577	10.193	43.31	56.50
21.975	0.505	8.760			12.59	14.06
21.725	0.552	9.858			4.33	3.29
21.625	0.568	10.262			1.70	-0.68
21.475	0.600	11.030			-3.96	-8.21

3.6.8 Comparison and Discussion

The geometrical and aerodynamic performance parameters of “DTU 10MW RWT” was used to design a blade in QBlade and the simulations were carried out. Similarly, the same case was also studied in ANSYS CFD. A grid independence study, domain sensitivity study along with placement of the geometry and different turbulence models were performed. CFD study showed that the pitch angle of the blade at higher velocities from 12 m/s to 25 m/s need to be reduced to achieve the expected results. These results are then compared to check the percentage of error (Table 3.6.9) between the three software. This

comparison concludes that the QBlade and CFD results for further design variants will be accurate and acceptable.

Table 3.6.9: Percentage of error between three software

	Percentage of error					
	(DTU vs CFD)		(CFD vs Qblade)		(DTU vs Qblade)	
Velocity (m/s)	C_t	C_p	C_t	C_p	C_t	C_p
5	5.74	-10.10	3.50	-5.19	-2.04	5.44
6	5.08	-1.22	1.62	3.28	-3.37	4.46
8	-1.13	0.61	-5.51	-1.21	-4.32	-1.83
9	-1.07	0.77	-5.07	-0.65	-3.94	-1.42
10	-1.24	0.62	-5.12	-0.59	-3.82	-1.21
11	-0.81	1.14	-4.42	0.12	-3.57	-1.01
12	5.07	2.93	-2.77	-3.36	-7.97	-6.38
16	1.18	0.00	-13.34	-9.20	-14.68	-9.20
20	2.42	2.21	4.11	-1.21	1.79	-3.45
25	-0.99	-0.09	0.68	-16.38	1.67	-16.28

Figure 3.6.12 shows the optimized thrust coefficient and the power coefficient obtained in CFD (current study), from the literature ("DTU 10 MW RWT" - EllipSys3D) and from QBlade. As mentioned previously, the QBlade software is an open source designed to calculate the wind turbine aerodynamics. An aerofoil can be designed and customized using this software because it is an integration functionality of the XFOIL/XFLR5. It will allow to design the required wind turbine blade and to study the aerodynamic performances. This software was used to design the additional case studies of the research; hence the baseline design was also compared with the QBlade results.

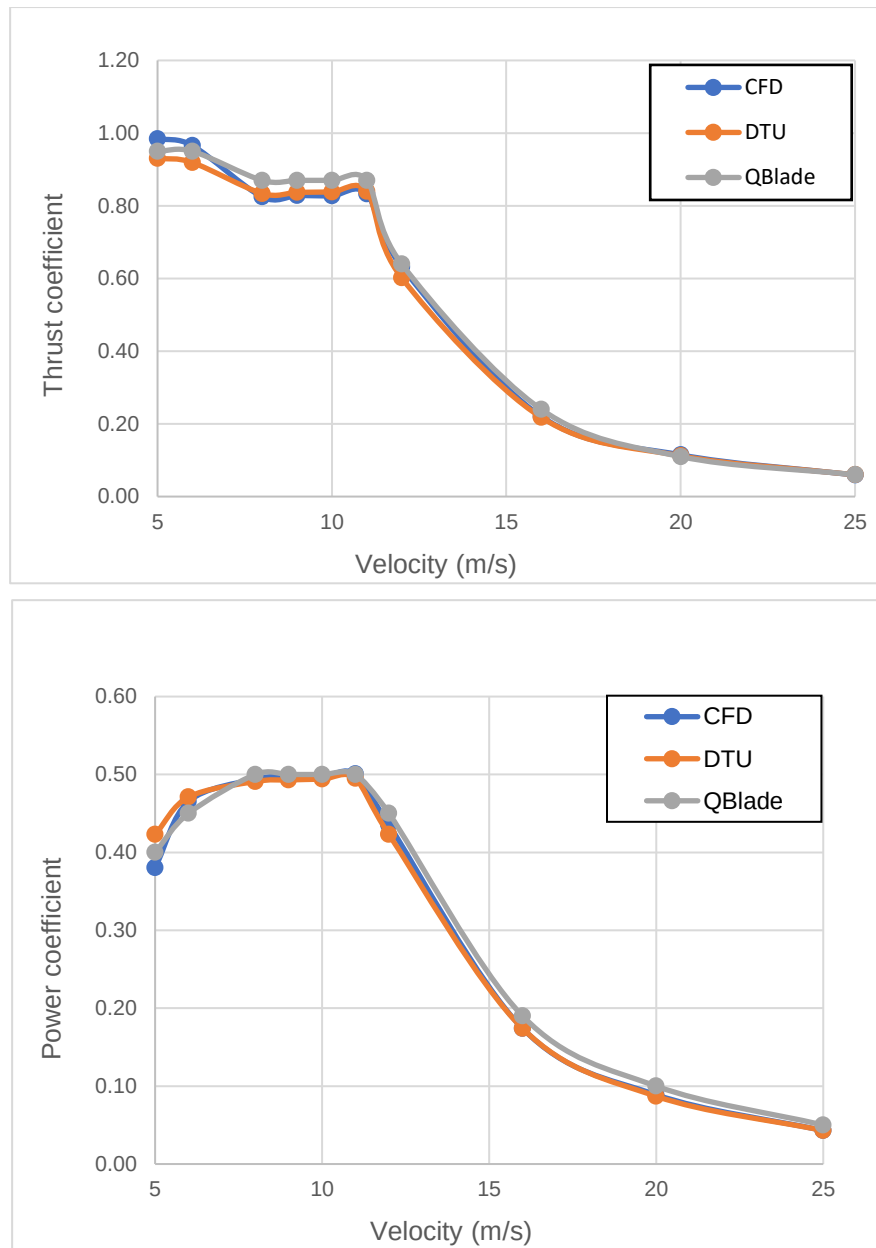


Figure 3.6.14: Thrust coefficient and power coefficient results comparison

4. Results and Discussion

4.1 Introduction

Across Europe, 5 to 6 *MW* wind turbines have been widely installed offshore, generating up to 500 *MW* power in a wind farm. The ‘Up Wind Research Project’ [122] supported by EU further considers the challenges that may arise to design a higher power generating wind turbine. For instance, a 20 *MW* wind turbine would require a very large rotor with an approximate diameter of 250 m and a tower height exceeding 150 m [122].

These large wind turbines will give rise to weight and load complications which are not desired and will need intense attention and further investigation while designing. Offshore wind farms with a typical size of 500 *MW* range employ 5 – 6 *MW* wind turbines, such wind farms are already operating across Europe [122]. Future trends envision the need for higher power generating wind turbines, at least 20 *MW* wind turbines, to be able to obtain annual energy production of 105 GWh [16].

Considering maintenance and operational costs due to very large amount of power being generated, any increase in size can be justified. The goal is however to design and manufacture fully functional wind turbines that are also cost effective.

4.2 Upscaling of Wind Turbines

When considering an upscaling approach to define the characteristics of an intended large scale offshore wind turbine, similarities are usually assumed with geometry and aerodynamic concepts of wind turbines, at constant tip speed between the baseline and the upscaled design. Through this approach structural properties and vital operations are established, this also helps determine the technological difficulties when upscaling.

When similarities with geometry are considered, weight and power scaling is done in accordance with $m \sim sf^3$ and $P \sim sf^2$. Here sf is the scaling

factor. Table 4.2.1 lists the properties of the IEA 5 MW wind turbine as a base model using a simple scaling approach [122] and it includes hub height, blade diameter and tip speed for 5 to 20 MW wind turbines.

Table 4.2.1: Properties for 5 to 20 MW wind turbine using simple scaling approach [122]

Power (MW)	Blade diameter (m)	Hub height (m)	Tip speed (m/s)
5	126	90	80
10	178	116	80
15	218	136	80
20	252	153	80

In the current research, the “DTU 10 MW reference wind turbine” has been upscaled to obtain a 20 MW wind turbine. Classical similarity rules have been applied to establish the final design, and the rotor area has been doubled to obtain twice the amount of power. Assuming similarities in geometry the following can be written for mass and power:

Mass m is scaled by:

$$m \sim sf^3 \quad (4.1)$$

Power P is scaled by:

$$P \sim sf^2 \quad (4.2)$$

with sf the geometric scaling factor:

$$sf = \sqrt{\frac{\text{Expected power after upscaling}}{\text{Power of existing upscalable model}}} = \sqrt{\frac{20}{10}} \quad (4.3)$$

4.3 Upscaling Methodology for Length Optimization

The baseline wind turbine blade has been upscaled to achieve 20 MW power using the above-described methodologies. Wind turbine blades with a larger span will produce more energy. The wind captured by the blades enables the blade to rotate; larger blades provide a wider area for the airflow to pass across, resulting in higher rotational power and force. However, the overall contribution made by increasing the blade span length as compared to other components, makes a real difference. Both the blade size and the velocity of the wind will determine the aerodynamic performance of the blade, this can be achieved using power/energy formulas. At a given swept area of the blade and velocity, the level of energy available in the airflow can be determined as:

$$E = \frac{1}{2} * mass\ flow\ rate * velocity^2 \quad (4.4)$$

Assuming a sea-level density of $1.23\ kg/m^3$, the mass flow rate can be determined as

$$mass\ flow\ rate = density\ (\rho) * velocity\ (v) * area\ of\ the\ blade\ (\pi r^2) \quad (4.5)$$

From the expression of mass flow rate (Equation 4.5), it is clear that the mass flow rate is directly proportional to the radius squared (r^2). Hence the longer the span the higher the mass flow rate, and therefore the higher the power/energy production.

For a large blade, a large nacelle is required, which therefore requires a large tower. Hence, blade length must be selected carefully to produce maximum power output and remain cost effective.

4.4 Upscaling the Blade Length Using QBlade

The blade was upscaled from the “DTU 10 MW RWT” to the 20 MW offshore wind turbine. In the current research the scope is given to the aerodynamic design only, even though structural design and aero-servo-elastic design are also important. The aerodynamic design is however key to improve the power output. The upscaled parameters for the 10 MW and 20 MW turbines

are listed in Table 4.4.1. The rotor diameter and hub height were upscaled using simple scaling approach. Rated wind speed is where the maximum power output is achieved. The minimum and maximum rpm are selected according to the results obtained from QBlade which uses BEM theory.

Table 4.4.1: Basic key parameters of 10 MW and 20 MW WT

Key parameters of wind turbines		
Parameters	DTU 10 MW RWT	20 MW WT
Rated power (MW)	10	20
Number of blades	3	3
Rotor placement	Upwind	Upwind
Rotor diameter (m)	178.3	252
Hub diameter (m)	5.6	8
Hub height (m)	119	167.9
Blade Length (m)	86.4	122
Rated wind speed (m/s)	11.4	12
Minimum rotor Speed (rpm)	6	4.4
Rated rotor speed (rpm)	9.6	9.2
Optimal TSR	7.5	9

4.4.1 Upscaled Blade Geometry

Using the geometrical parameters in Table 4.4.2, the blade geometry (Figure 4.4.1) has been redesigned in QBlade. The length of the blade is increased to 122 m with a hub radius of 4 m. The blade is placed at the centre of the hub, henceforth the radius of the blade is 126 m. At this stage of the work, the chord length, twist, and thread axis of the blade are not modified, and they remain identical to the baseline model. The location of the aerofoils along the span has been placed according to the r/R values. Where $R = 126$ m and r is the radial stations. The optimization procedure will highlight the effects of extending the blade length.

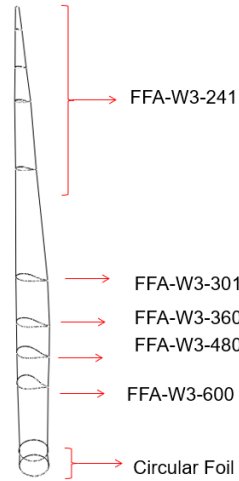


Figure 4.4.1: Blade geometry for upscaled blade

Table 4.4.2: upscaled wind turbine blade geometry parameters

Aerofoils	Radius r (m)	r/R	Chord (m)	Twist (degree)	Thread Axis in [% chord]
Cylinder	0.00	0.00	5.38	0.00	0.50
	3.91	0.03	5.38	14.50	0.50
FFA-W3-600	15.50	0.12	5.45	14.43	0.44
	23.81	0.19	5.87	12.54	0.40
FFA-W3-480	29.48	0.23	6.11	9.96	0.37
FFA-W3-360	36.67	0.29	6.20	7.83	0.35
	43.22	0.34	6.09	6.72	0.35
FFA-W3-301	51.28	0.41	5.79	5.62	0.35
FFA-W3-241	71.19	0.57	4.70	2.89	0.35
	84.92	0.67	3.87	0.90	0.35
	101.93	0.81	2.90	-1.14	0.35
	119.95	0.95	1.90	-2.95	0.35
	126.00	1.00	0.60	-3.43	0.35

The maximum C_p of 0.467 for the current blade was obtained at a tip speed ratio of 9 and a pitch angle of 0 *degree*, as shown in Figure 4.4.2. As mentioned previously, 100% extraction of energy from the wind is not possible as the energy is extracted from the wind by slowing it down and removing part of its kinetic energy. As the wind is slowed down, the air behind the blade will be slower. If further kinetic energy is extracted, the downstream air will block the next intake of air which will hit a wall of stalled air.

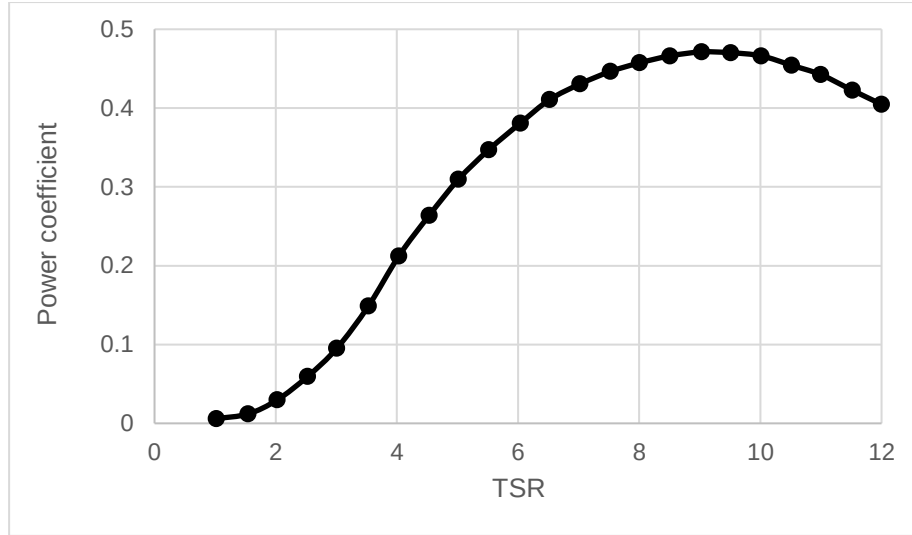


Figure 4.4.2: Power coefficient vs tip speed ratio for an upscaled blade

4.4.2 Results

The upscaled blade was simulated for different rotational speeds and pitch angles. The simulation parameters used to calculate the power and thrust are $\rho = 1.225 \text{ kg/m}^3$, $\mu = 1.784 \times 10^{-5} \text{ kg/(m.s)}$. The velocity range between 5 m/s and 25 m/s, where the rated power output was achieved at 12 m/s. Velocities selected to perform the simulations and to study the flow physics are marked with a * symbol in Table 4.4.3. These velocities* are selected due to their corresponding rotational speed being unique, and to reduce the computational resources used in the research. These results are then compared with the results obtained with CFD in FLUENT.

The rotational speed was varied between 4 and 10 rpm and the optimized rotational speed was selected at a fixed pitch angle of zero. However, at high wind speeds, between 12 to 25 m/s, the pitch angle zero did not provide the required power and thrust results, hence, a pitch angle optimization was carried out. To understand the problem better, for a pitch angle of 0 degree and a velocity of 12 m/s, a 20 MW power was achieved at a rotational speed of 5.4 rpm. This scenario is however not plausible because the wind flowing through the blades rotates them, and at a speed of 11 m/s, the rotational speed is 8 rpm. Hence, for a wind speed of 12 m/s the rotational speed should be higher than 8 rpm.

To achieve the optimum power and thrust results the pitch angle was varied between 0 and 20 *degree*. The rotational speed and pitch angle were varied, and the optimum rotation speed and pitch angle were selected to achieve 20 MW power. Table 4.3.3 provides the detailed aerodynamic results and the operational parameters, based on QBlade, at different velocities.

Table 4.4.3: Operational parameters and aerodynamic results at different velocities (the * symbols indicate the velocities to be simulated in CFD)

Velocity (m/s)	Rotational speed (RPM)	Pitch angle (degree)	Power (MW)	C_P	Thrust (MN)	C_t
5*	4.4	0.00	1.59	0.416	0.690	0.903
6*	5.0	0.00	2.90	0.439	0.989	0.899
7	5.0	0.00	4.80	0.458	1.170	0.782
8*	5.4	0.00	7.29	0.466	1.520	0.777
9*	6.4	0.00	10.40	0.467	1.920	0.776
10*	7.0	0.00	14.27	0.467	2.390	0.782
11*	8.0	0.00	19.00	0.467	2.920	0.790
12*	9.2	3.96	20.20	0.383	2.500	0.568
13	9.2	6.00	20.20	0.301	2.200	0.426
14	9.2	7.50	20.20	0.241	1.980	0.331
15	9.2	8.80	20.20	0.196	1.800	0.262
16*	9.2	9.90	20.20	0.161	1.695	0.217
17	9.2	11.00	20.20	0.135	1.600	0.181
18	9.2	12.05	20.20	0.113	1.510	0.153
19	9.2	13.00	20.20	0.096	1.480	0.134
20*	9.2	13.91	20.20	0.083	1.405	0.115
21	9.2	14.82	20.20	0.071	1.390	0.103
22	9.2	15.70	20.20	0.062	1.320	0.089
23	9.2	16.55	20.20	0.054	1.290	0.080
24	9.2	17.39	20.20	0.048	1.260	0.072
25*	9.2	18.18	20.20	0.042	1.240	0.065

These aerodynamic results are compared with those from the “DTU 10 MW RWT” to check both the pattern and agreement. Figure 4.4.3 shows the power and thrust versus the velocity. As can be seen, the trends of both the Power and thrust curves obtained for 20 MW with QBlade agree well with the 10 MW wind turbine blade-based baseline results obtained using EllipSys3D. Till the rated velocity of 12 m/s, the power increases linearly and remains constant afterwards until it reaches the cut-out velocity. In the region between rated and

cut-out velocity, a constant power of 20.2 MW is achieved for the 'new' wind turbine. Henceforth, the ideal power curve is achieved from both the variable rotational speed and the variable pitch angle configuration used in this research.

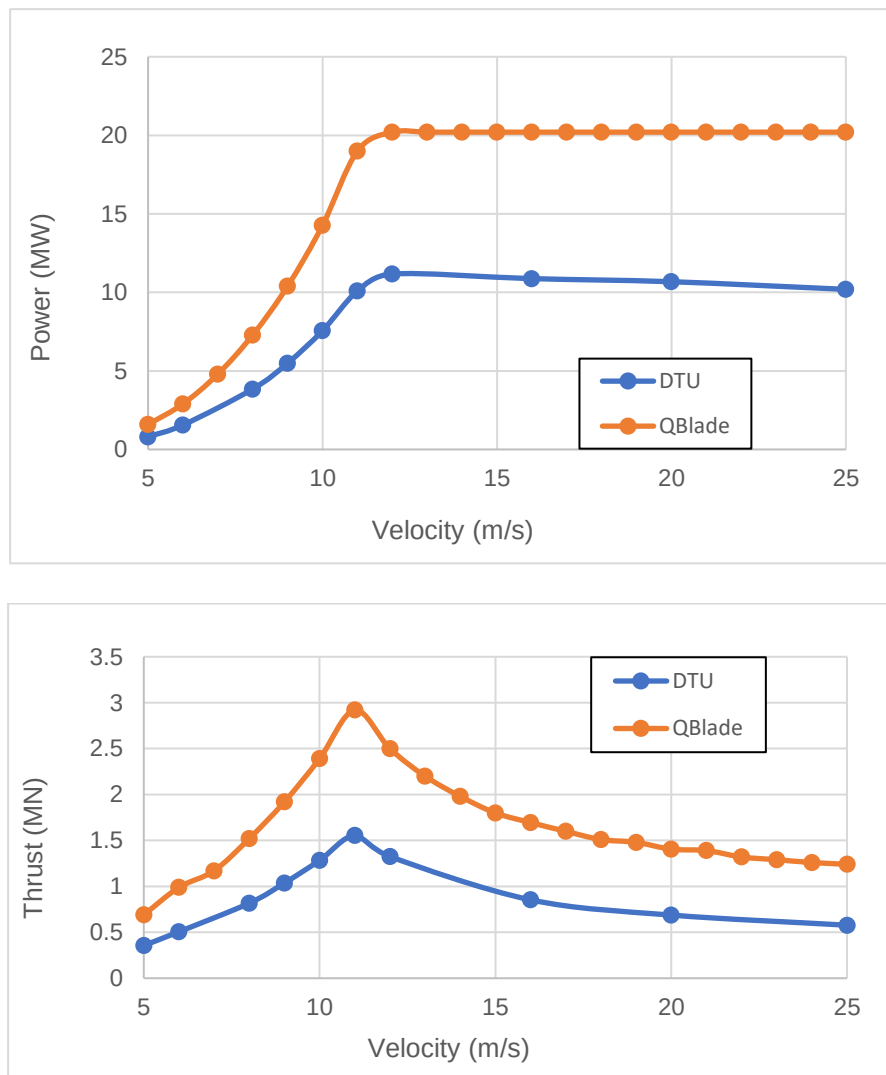


Figure 4.4.3: Power and thrust versus velocity plot

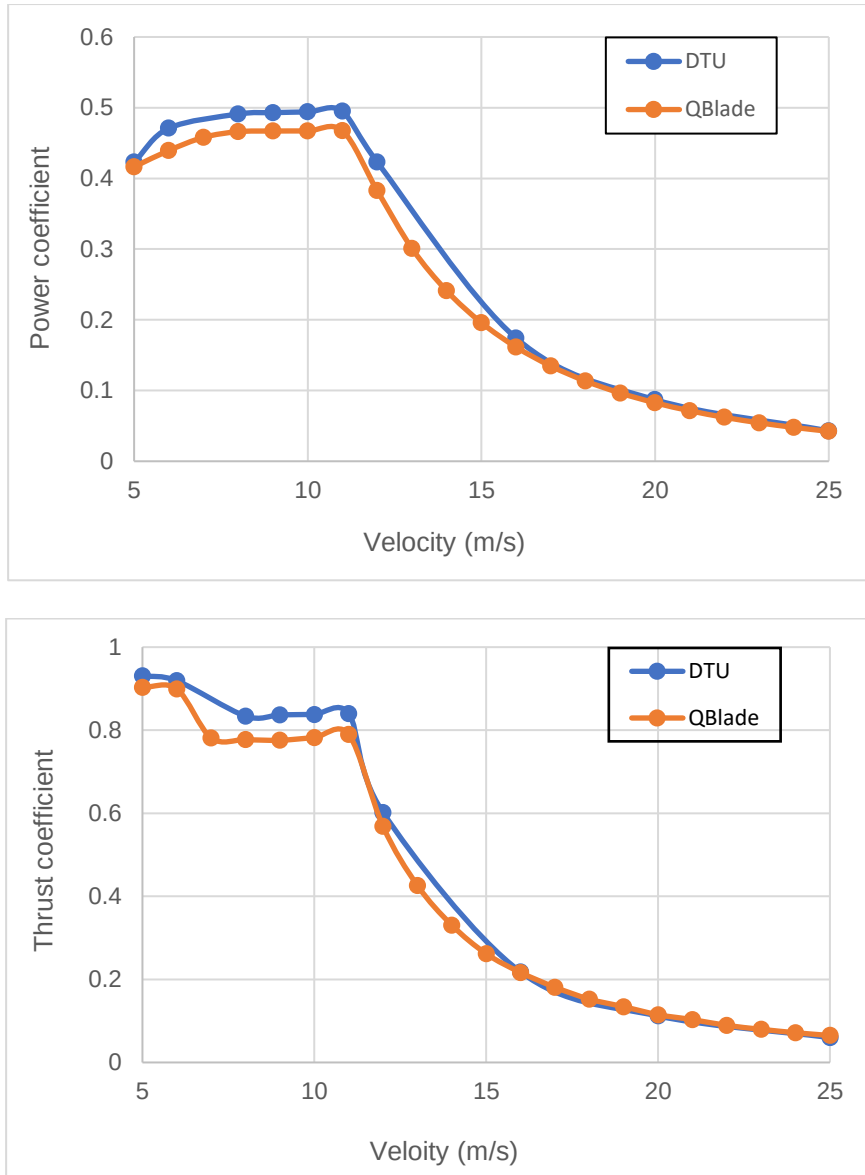


Figure 4.4.4: Power coefficient and thrust coefficient versus velocity plot

The maximum thrust of 2.92 MN in Figure 4.4.3 was achieved for the ‘new’ design at a velocity of 11 m/s . The thrust increased from 1.56 MN for the 10 MW turbine to 2.92 MN for the 20 MW turbine at this velocity, and the corresponding power increased from 11.1 MW to 20.2 MW at 12 m/s . This shows a 87.78% increase in thrust and 80.83% increase in power with the ‘new’ design compared to the “DTU 10 MW RWT”.

The geometrical design and operational conditions obtained from QBlade are then taken into the ANSYS CFD software for further studies and comparison of the aerodynamic results.

4.5 Upscaled Blade Length Study Using CFD

The geometrical details described in the previous section are exported from QBlade to build a design model in ICEM CFD. Figure 4.5.1 shows the 3 bladed upwind upscaled geometry. Note that a single blade is selected for the flow simulations, this will reduce computational resources being used, and therefore the computational cost and time.



Figure 4.5.1: Upscaled blade geometry

4.5.1 Meshing

A hexa O-grid mesh was generated around the blade and in the domain immediately surrounding the blade. The outer domain size built for this case was 3D upwind, 5D downwind, and 4D long on the top of the blade, with $D = 252\text{ m}$ being the diameter of the blade. The inner domain was 1.5R long upwind, downwind and on the top surface, with R being the radius of the blade. The domain was created and meshed similarly to the baseline study described previously in Chapter 3, Figure 3.6.3. The mesh on the single blade, for a section of the blade, and around the aerofoil section are shown in Figure 4.5.2.

The blade and domain mesh were then imported to the flow solver ANSYS FLUENT to start the flow simulation. Note that the finalized boundary conditions

considered, the domain sensitivity and placement of the blade carried out previously for the baseline study and described in Chapter 3 are considered valid and used here without modifying any of the parameters reported from Table 3.6.4.

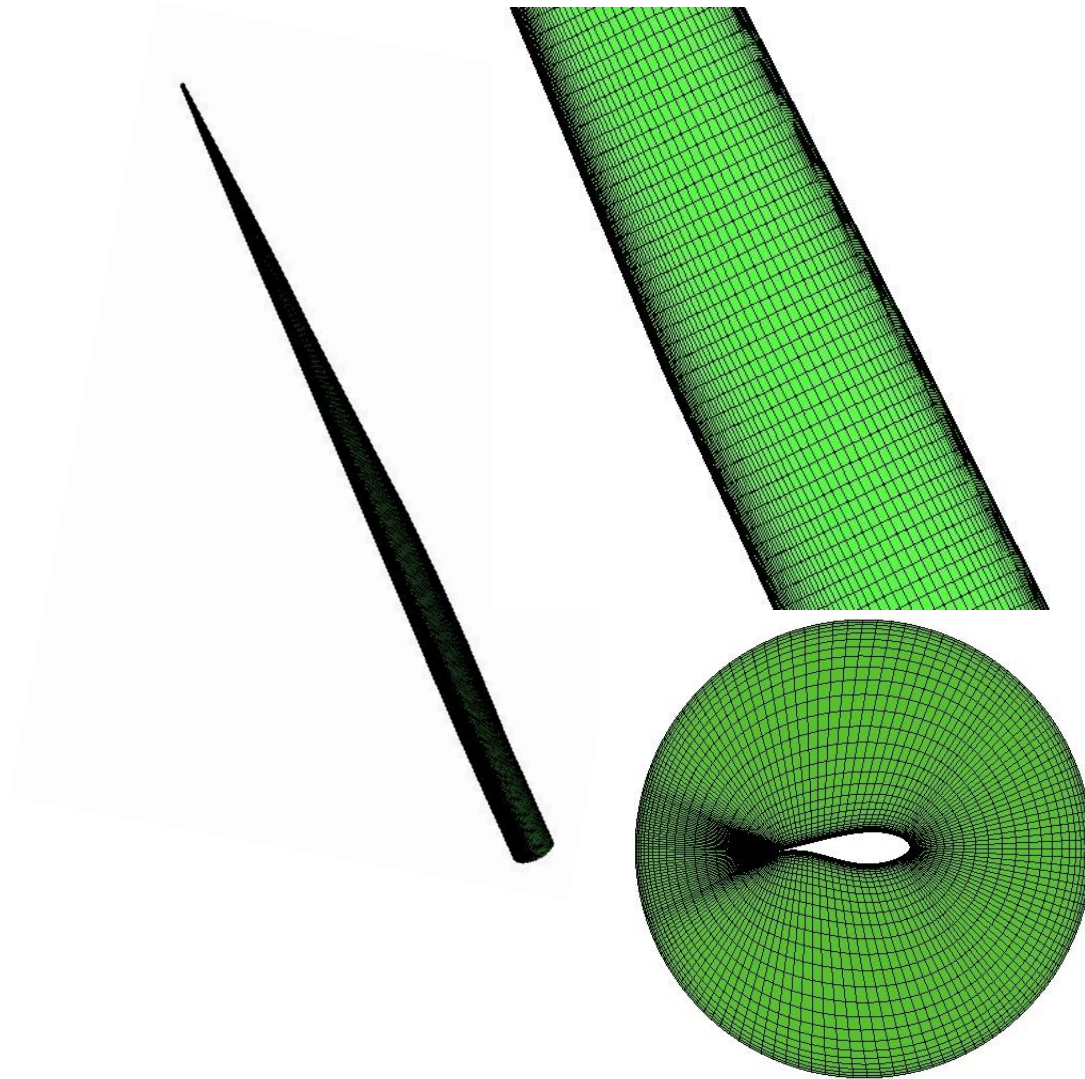


Figure 4.5.2: Mesh on blade geometry

4.5.2 Results

The optimized rotational speed was taken from QBlade, and no variation was made in FLUENT. However, the pitch angle was reduced to achieve the rated power output. Table 4.5.1 shows the pitch angle reduction carried out to achieve the rated power of 20 MW.

The angle of the pitch was reduced for the highest velocities, between 12 m/s and 25 m/s . For the length optimization case study, the pitch angle for the lowest velocities was zero and the results agreed well with the QBlade results. However, because of the pitch angle sensitivity study, the results of velocity with a pitched blade did not match well with QBlade.

Table 4.5.1: Pitch angle optimization for higher velocities

Velocity (m/s)	Original Pitch angle (degree)	Reduced pitch angle by degree	Optimized pitch angle by degree	Optimized pitch angle (degree)
12	3.96	1	1	2.96
16	9.9	1, 1.25	1.25	8.65
20	13.91	1.25, 1.35	1.35	12.56
25	18.18	1.25, 1.35, 1.5	1.5	16.68

The first simulation was carried out for the reduced pitch angle itself. From the baseline study it was concluded that the minimum reduction in pitch angle required for operating at a velocity of 12 m/s was 1 *degree*, hence for higher velocities, the pitch angle reduction was simulated progressively from the previously optimized pitch angle reduction degree. For instance, in the baseline study for 20 m/s velocity, the optimized pitch angle reduction was of 1.25 *degree*, hence the first simulation of the length optimization was carried out from a 1.25 *degree* reduction. In this section velocities 16 m/s and 25 m/s are shown and remaining velocities results are in Appendix 7.3

Table 4.5.2 shows the aerodynamic results for a velocity of 16 m/s at different pitch angles. The optimized pitch angle for 16 m/s is 8.65 *degree* at a reduction of 1.25 *degree* with an error -0.79% and -3.19% against QBlade results for the thrust and power, respectively. These percentage of error are acceptable at this stage of the study because further improvement of these results will be carried out in next design variant i.e., chord and twist optimization.

Table 4.5.2: Pitch angle sensitivity study for velocity 16 m/s

	Pitch angle sensitivity study (CFD)		QBlade results		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
8.90	1.574	18.123	1.695	20.20	-7.12	-10.28
8.65	1.681	19.554			-0.79	-3.19

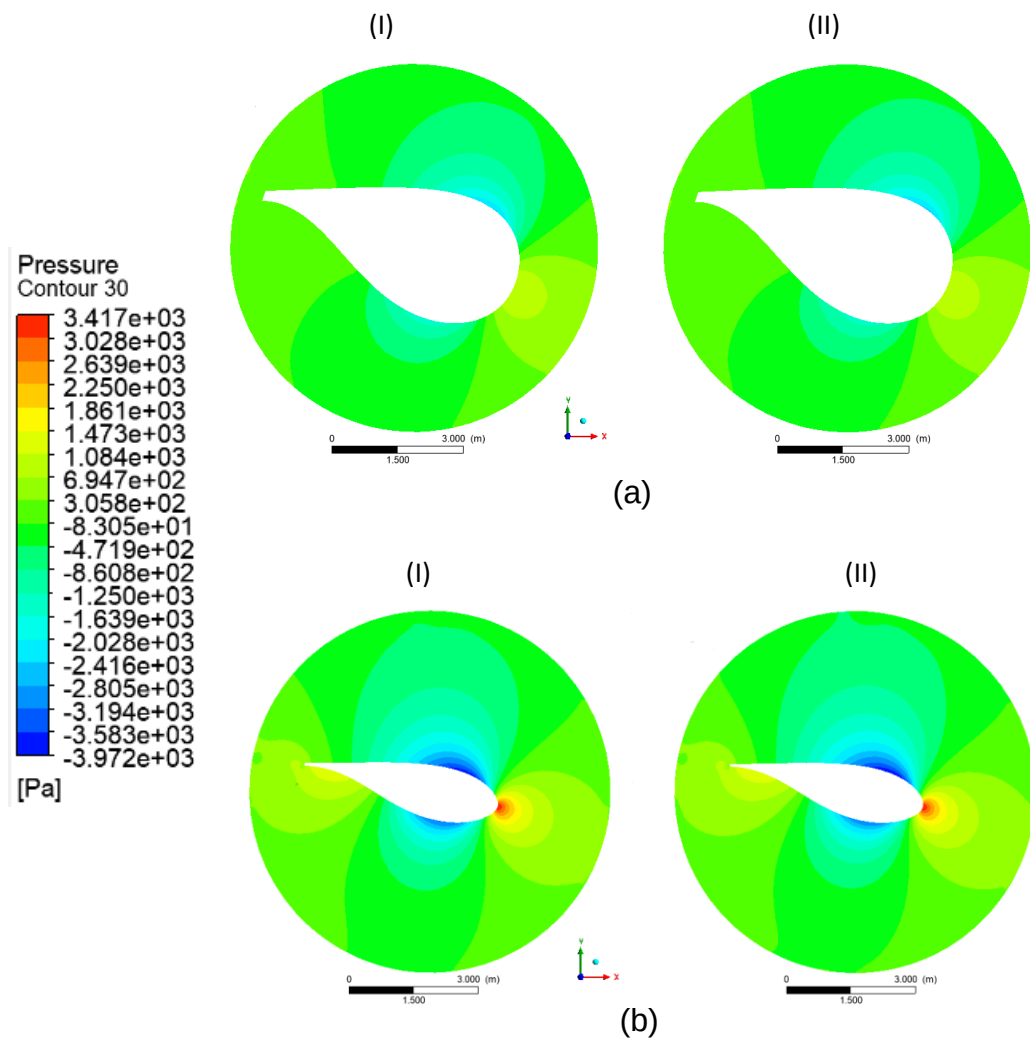
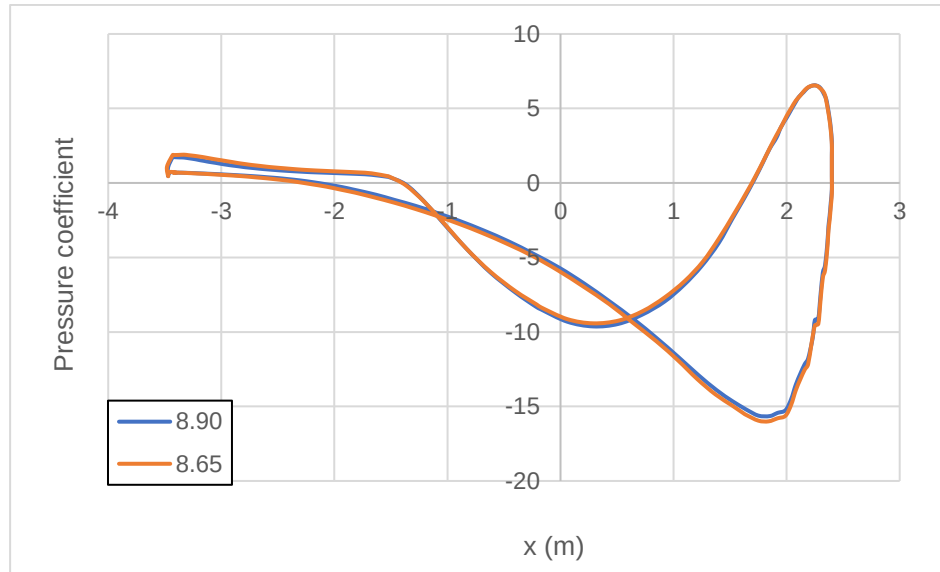


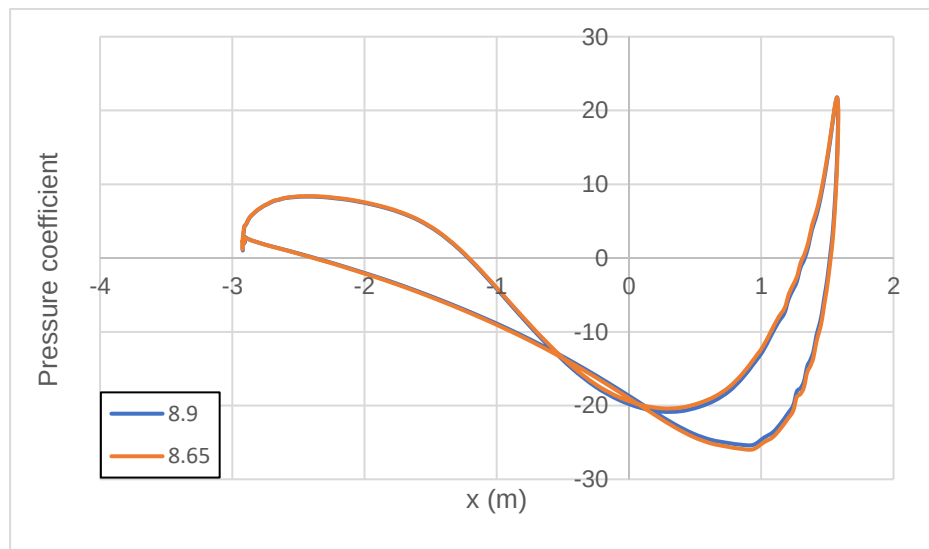
Figure 4.5.3: Pressure contour plots for pitch angle (I) 8.9° and (II) 8.65° at (a) 30% and (b) 60% of the span

The pressure contour plots and pressure coefficient C_p plots are shown in Figures 4.5.3 and 4.5.4, respectively. These plots have been taken at 30% and 60% of the blade span for both pitch angles at a velocity of 16 m/s. There is no much difference between the pitch angle reduction in terms of contour plots, but

the slight difference seen for C_p produces an error between 6% and 7% with respect to thrust and power.



(a)



(b)

Figure 4.5.4: Pressure coefficient plots for pitch angle 8.9 and 8.65 at (a) 30% and (b) 60% of the span

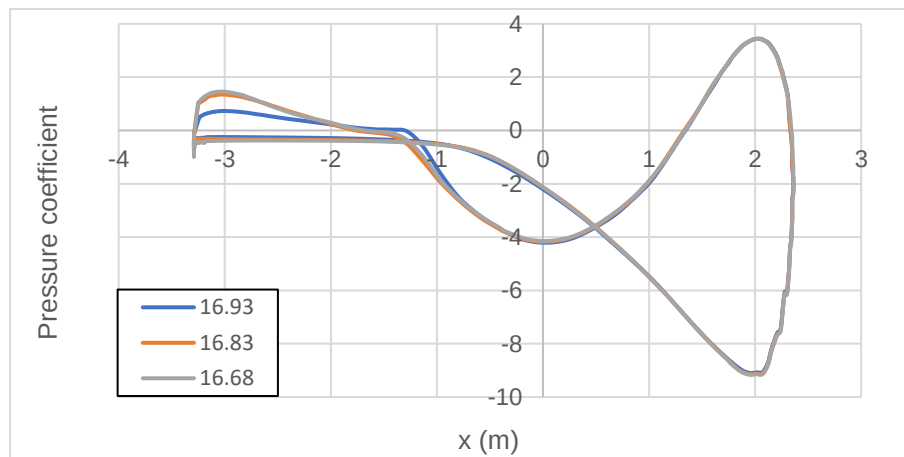
Table 4.5.3 provides the detailed aerodynamic results obtained from the pitch angle reduction for 25 m/s. From the optimized pitch angle i.e., 16.68 degree (1.5 degree reduction), the errors for thrust and power are −10.98% and −5.82%, respectively. From those results it can be concluded that

the required thrust and power are not obtained. However, these results can be improved by carrying out a chord and twist optimization procedure, this will be the next stage of the study.

Table 4.5.3: Pitch angle sensitivity study for velocity 25 m/s

Pitch angle (degree)	Pitch angle sensitivity study (CFD)		QBlade results		Percentage of error	
	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
16.93	0.997	16.294	1.24	20.2	-19.62	-19.33
16.83	1.046	17.557			-15.64	-13.08
16.68	1.104	19.023			-10.98	-5.82

Figures 4.5.5 and 4.5.6 show the pressure contour plots and pressure coefficient C_p for a velocity of 25 m/s at different pitch angles. Similarly, to the 16 m/s velocity, the plots are taken at 30% and 60% span of the blade. The thrust and power were increased by 8.64% and 13.51% compared to the pitch angles of 16.93 degree and 16.68 degree.



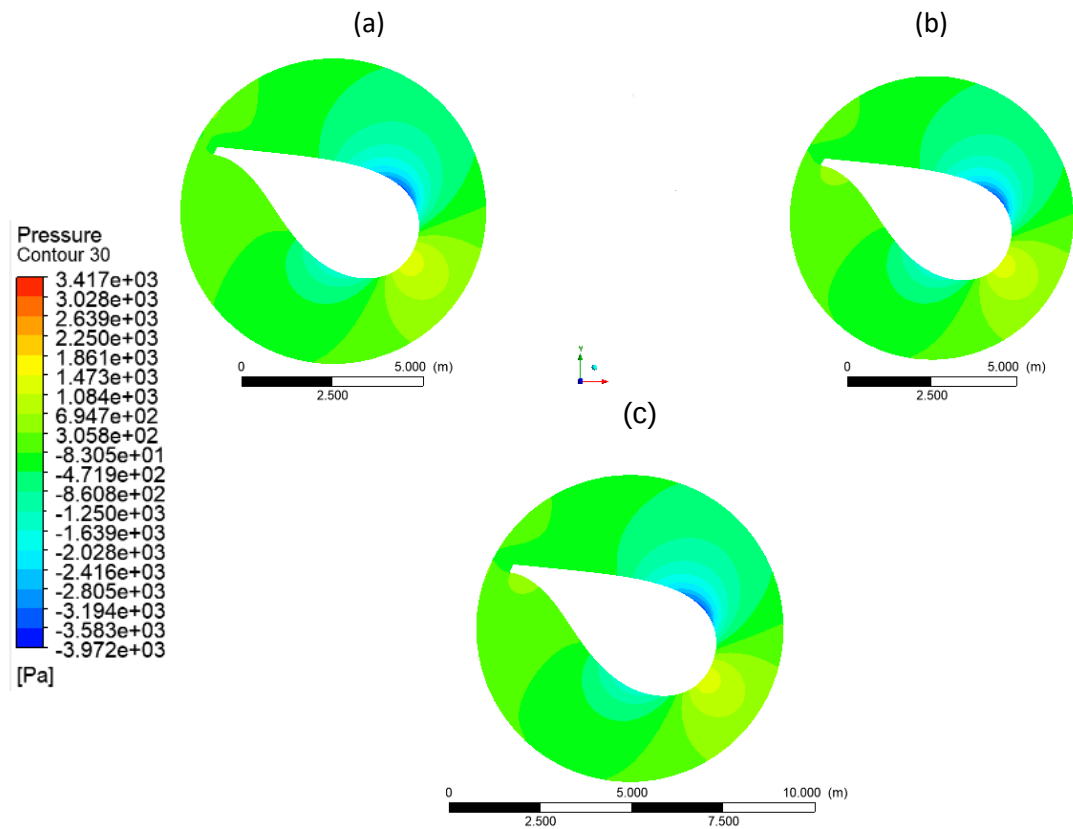
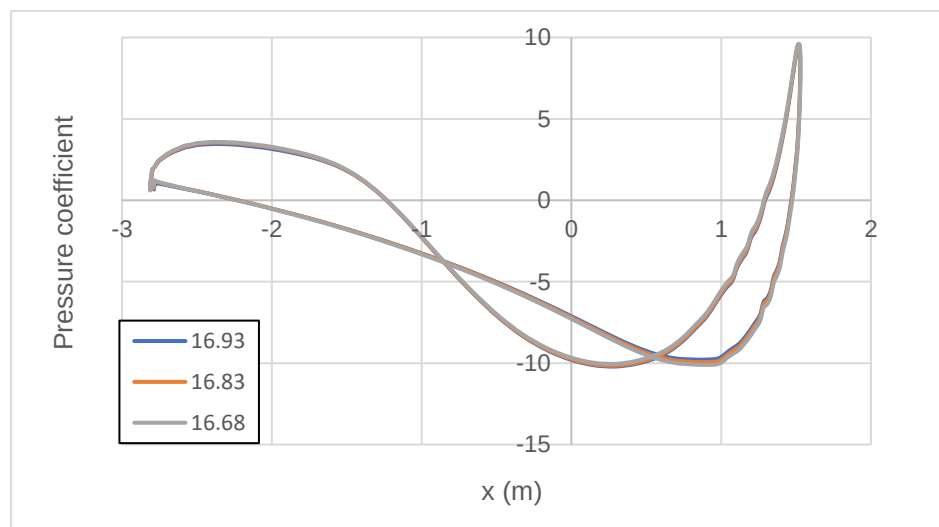


Figure 4.5.5: Pressure coefficient C_p and pressure contour plots for velocity 25 m/s for different pitch angle (a) 16.93 degree (b) 16.83 degree (c) 16.68 degree at span 30%



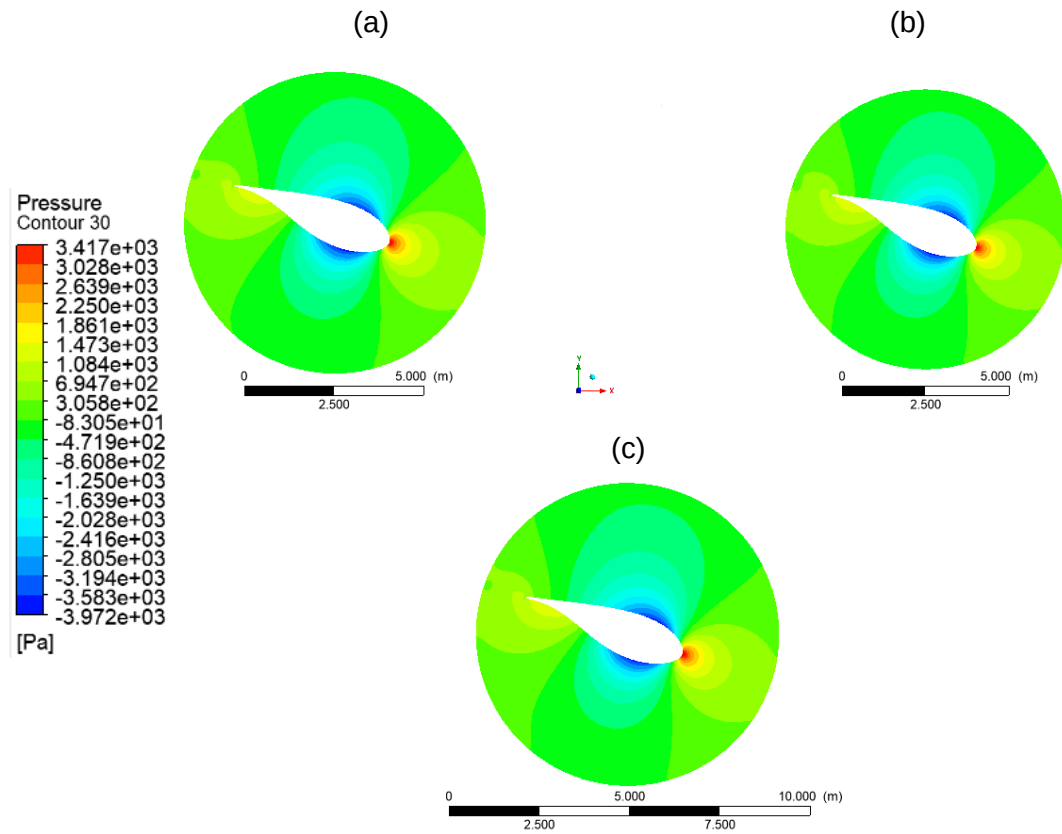
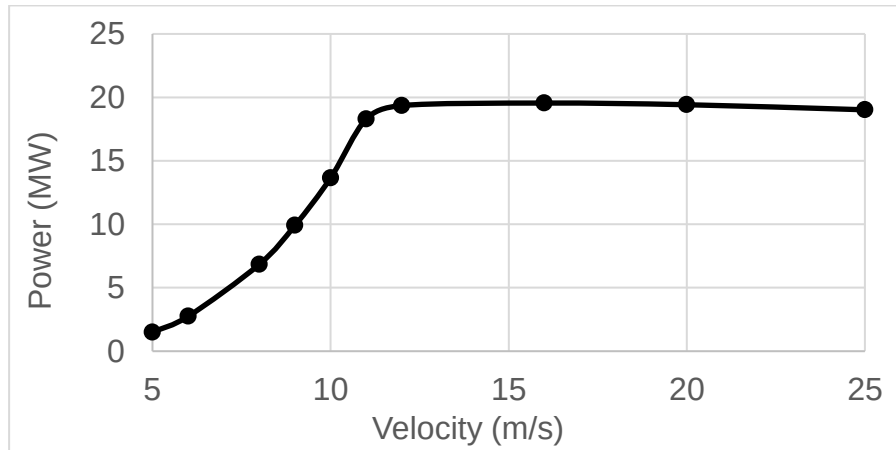
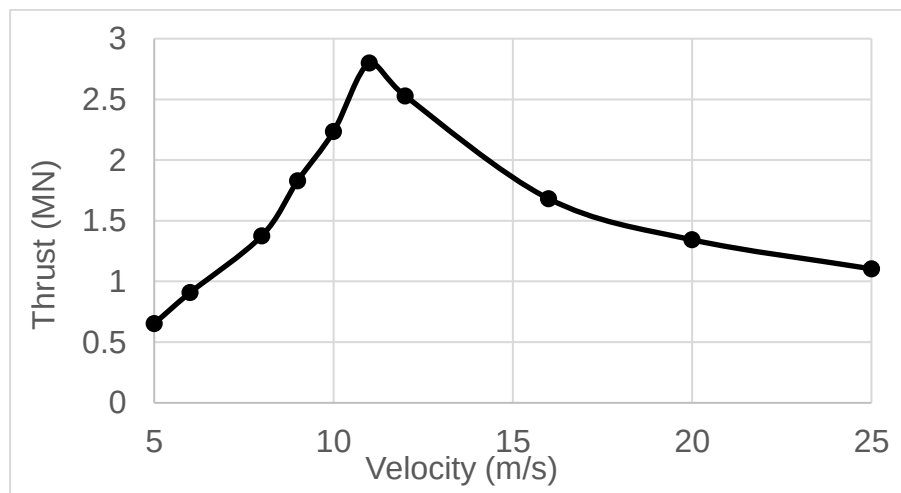


Figure 4.5.6: Pressure coefficient C_p and pressure contour plots for velocity 25 m/s for different pitch angle (a) 16.93 degree (b) 16.83 degree (c) 16.68 degree at span 60%

The results of the CFD study regarding the length optimization are shown in Figure 4.5.7, and Table 4.5.4. Similarly, to QBlade results, the CFD results also follow the ideal power and thrust curves. A maximum power of 19.55 MW is achieved at 16 m/s whereas a maximum thrust of 2.79 MN is achieved at 11 m/s. In the power curve, the rated power is achieved from the rated velocity of 12 m/s to the cut-out velocity of 25 m/s.

Table 4.5.4: Aerodynamic results from CFD for length optimization

Velocity (m/s)	Rotational speed (RPM)	Pitch angle (degree)	Thrust (MN)	C_t	Power (MW)	C_P
5	4.4	0.00	0.653	0.855	1.50	0.39
6	5.0	0.00	0.908	0.825	2.74	0.42
8	5.4	0.00	1.376	0.704	6.85	0.44
9	6.4	0.00	1.829	0.739	9.91	0.45
10	7.0	0.00	2.235	0.732	13.65	0.45
11	8.0	0.00	2.798	0.757	18.30	0.45
12	9.2	2.96	2.528	0.575	19.36	0.37
16	9.2	8.65	1.681	0.215	19.55	0.16
20	9.2	12.56	1.343	0.110	19.42	0.08
25	9.2	16.68	1.104	0.058	19.02	0.04



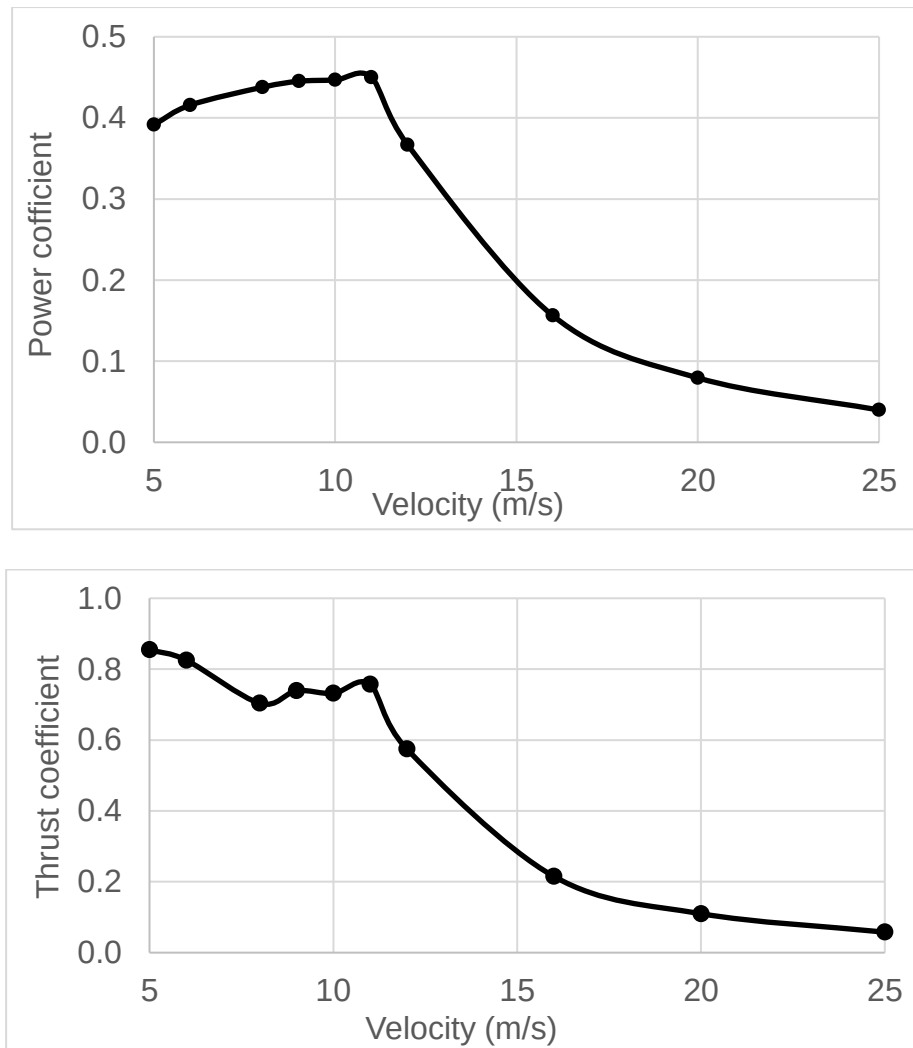


Figure 4.5.7: Aerodynamic results from CFD study

4.5.3 Discussion

In the current case, the span of the blade was upscaled from 89.166 *m* to 126 *m*. The aim of this case study was to increase the power from 10 *MW* to 20 *MW*. The geometry of the blade was designed in QBlade to obtain 20 *MW* power output with variable rotation speed and variable pitch angle. The optimized power and thrust results were then compared with the baseline results. The power was increased by 80.84% and the thrust was increased by 88.67% at a rated velocity of 12 *m/s*. From these comparisons it could be concluded that the blade geometry parameters and aerodynamic performance parameters were suitable for further CFD investigations.

The blade geometry was then re-designed and meshed in ICEM CFD, and flow simulations carried out in FLUENT. Results at high velocity with a pitch

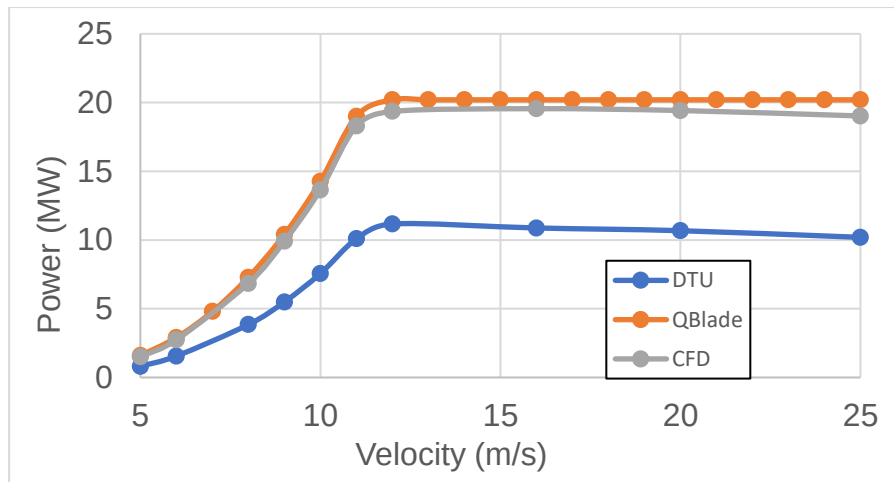
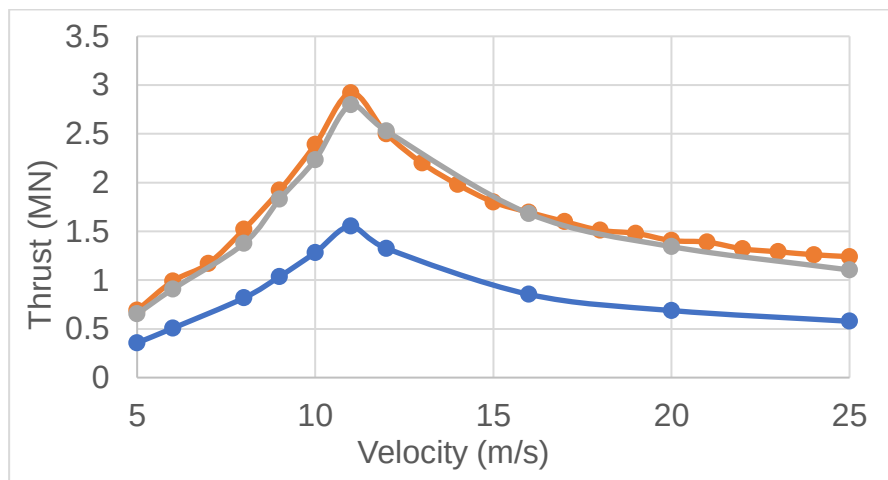
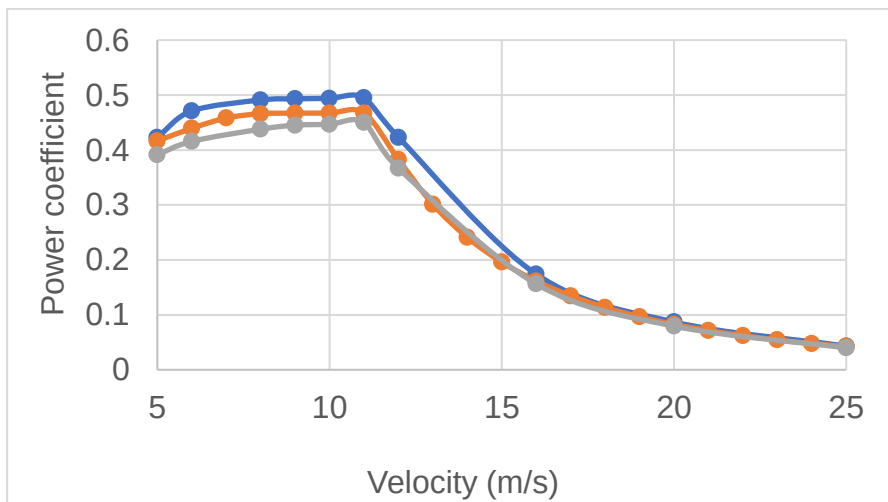
angle did not agree well with the QBlade results due to the flow sensitivity. Hence the pitch angle of the highest velocities from 12 m/s to 25 m/s were reduced to obtain the required results. This optimization increased the power by 73.32% and the thrust was increase by 90.79% when compared to the baseline study.

Table 4.5.5: Percentage of error between QBlade and CFD results

Velocity	Percentage of error	
	C_t	C_p
5	-5.39	-5.95
6	-8.22	-5.41
8	-9.50	-6.05
9	-4.76	-4.67
10	-6.49	-4.36
11	-4.17	-3.71
12	1.12	-4.17
16	-0.80	-3.20
20	-4.40	-3.85
25	-10.98	-5.83

The percentage of error between QBlade and CFD results are summarised in Table 4.5.5. The percentages of errors have been compared in terms of power and thrust coefficients. The maximum errors of thrust and power coefficients were -10.98% and -6.05% , respectively. Therefore, it can be concluded that the CFD simulations are underpredicting the results when compared to QBlade. These errors are due to the turbulence model used in the CFD approach, and due to viscosity effects. QBlade simulations are indeed non-viscous simulations and are based on the blade element momentum theory (BEM) whereas CFD simulations are Reynolds-averaged Navier–Stokes equations (RANS) based with the $k-\omega$ SST turbulence model. It is thus expected that CFD-based results provide a slightly better accuracy than QBlade-based results.

Figure 4.5.8 shows the comparison between the baseline, QBlade and CFD results. The pattern of the results matches well with each other and the results of QBlade and CFD agrees well with each other.



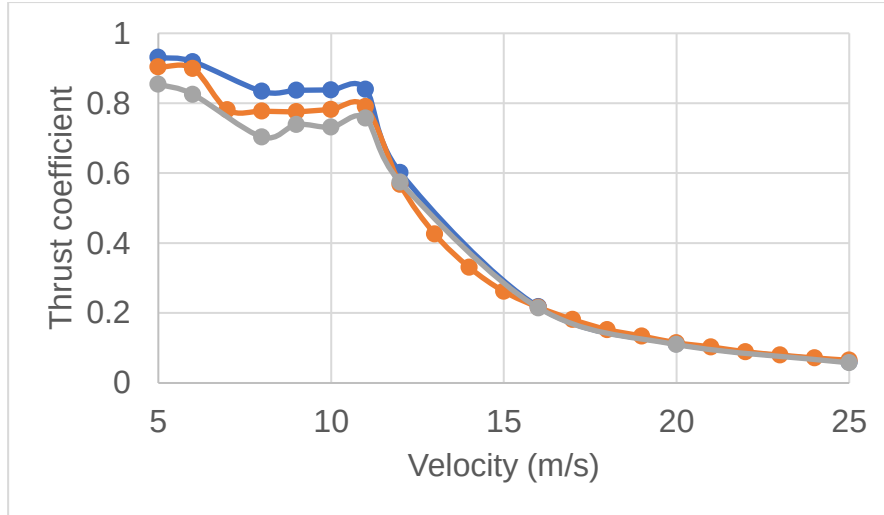


Figure 4.5.8: Aerodynamic results comparison between baseline (“DTU 10 MW RWT”), QBlade, and CFD

4.6 Upscaling Methodology for Chord Length and Twist Angle Optimization

A wind turbine blade is a combination of N number of aerofoils, which produce lift and drag at an angle of attack. Hence the aerodynamic results of the wind turbine can also be calculated in terms of lift and drag along with power and thrust. The blade element theory concept will help to calculate the required chord and twist of the wind turbine blade design. To simplify the calculation some assumptions are made. Firstly, the aerodynamic interactions between the aerofoils are neglected. Secondly, all aerodynamic results for these aerofoils are calculated in terms of lift and drag.

The lift acts perpendicular and the drag acts parallel to the free stream velocity of an aerofoil. In 2D, the free stream velocity hits the aerofoil’s leading edge and produces aerodynamic results, depending on the angle of attack. But in 3D, the relative velocity hits the leading edge of the blade and as a result the power and thrust at different pitch angles are obtained, this is determined by the velocity triangle as shown in Figure 4.6.1.

The relative wind velocity v_{rel} is a vector sum of the wind velocity v_1 at the wind turbine blade and the angular velocity Ω at the rotor distance r i.e.,

$$v_{rel} = \sqrt{(v_1^2 + (\Omega r)^2)} \quad (4.6)$$

Note that the angular velocity Ω must be in revolutions per second

Here, α is the angle of attack, θ_T is the angle of twist, $\theta_{p,0}$ is the blade pitch angle at the tip, φ is the angle of relative velocity. The angles between the plane of the blade rotation and the chord line are the section pitch angle θ_p . The incremental forces are: dF_T - tangential force, dF_N - normal force, dF_D - drag force, and dF_L - lift force. The torque is produced by the tangential force.

The blade twist can be calculated with the help of the blade pitch angle and the section pitch angle i.e.,

$$\theta_T = \theta_p - \theta_{p,0} \quad (4.7)$$

Similarly, the angle of relative velocity φ can be calculated as,

$$\varphi = \theta_p + \alpha \quad (4.8)$$

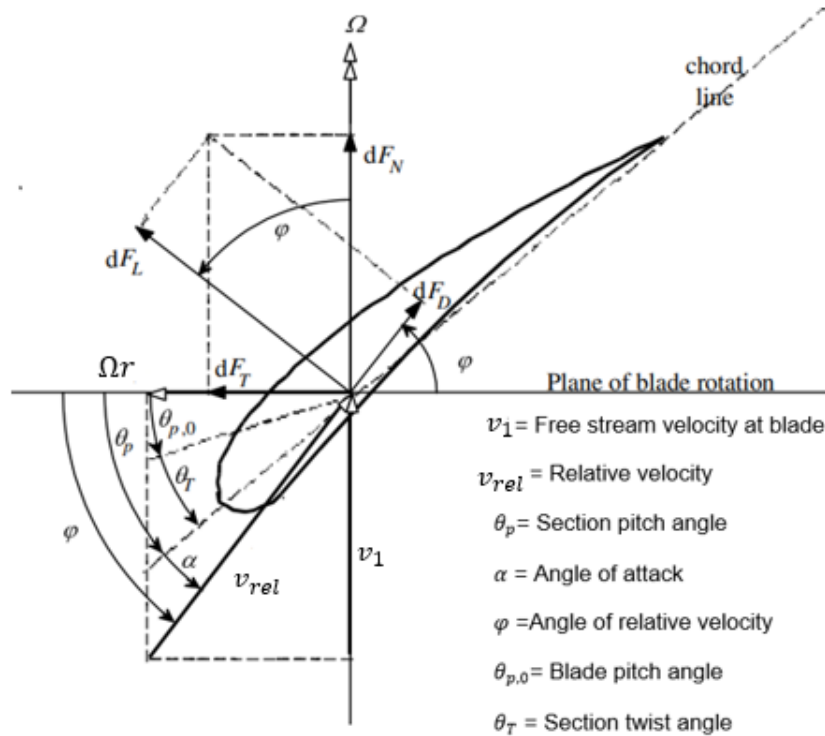


Figure 4.6.1: Aerodynamic forces, velocities, and angles at blade tip [123]

The shape of the blade and its performance can be determined with the blade element theory and the momentum theory. If the axial induction factors a and tangential induction factor a' are accounted for, the free stream velocity v_1 will be $v_1 (1 - a)$ and the angular velocity Ωr will be $\Omega r (1 + a')$. Therefore, the angle of relative velocity can be calculated as

$$\tan \varphi = \frac{v_1(1 - a)}{\Omega r (1 + a')} \quad (4.9)$$

Similarly,

$$v_{rel} = \frac{v_1(1 - a)}{\sin \varphi} \quad (4.10)$$

Few assumptions are made to simplify the blade element theory and the momentum theory, and a realistic shape of the blade can be achieved. The assumption would be the axial induction factor a to be $1/3$, and the wake rotation being neglected, hence, a' is zero, and there are no losses.

Now rewriting Equation 4.9 with known variables as:

$$\tan \varphi = \frac{v_1(1 - 1/3)}{\Omega r (1 + 0)} \quad (4.11)$$

Therefore,

$$\varphi = \tan^{-1} \left(\frac{2}{3\lambda_r} \right) \quad (4.12)$$

Here,

$$\lambda_r = \lambda \frac{r}{R} \quad (4.13)$$

Where,

r : Radius at a distance along the span of the blade

R : Blade radius

and Equation 4.10 can be rewritten as:

$$v_{rel} = v_1 (1 - \frac{1}{3}) / \sin \varphi \quad (4.14)$$

Therefore,

$$v_{rel} = \frac{2}{3} v_1 / \sin \varphi \quad (4.15)$$

The chord C of the blade can now be calculated from the BEM theory

$$C_L B C / 4\pi r = \tan \varphi \sin \varphi \quad (4.16)$$

Rearranging Equation 4.16

$$C = \frac{8\pi r \sin \varphi}{3 B C_L \lambda_r} \quad (4.17)$$

4.7 Upscaling Blade Chord Length and Twist Angle Using QBlade

To continue with the calculation, some parameters need be known beforehand for the current design, they are:

- tip speed ratio (λ) = 7.5
- number of blades (B) = 3 and
- radius (R) = 126 m

The lift coefficients C_L and the drag coefficient C_D of the current blade's aerofoils have been studied in this research in Chapter 3, Section 3.5, hence C_L , C_D , at different angle of attack (α) are known. To obtain the chord and twist of the blade which can produce the power within the Betz limit, one must know the minimum drag coefficient to lift coefficient ratio (C_D/C_L).

From Equations 4.7 and 4.17, the twist and chord for the current design have been calculated as provided in Table 4.7.1.

Table 4.7.1: Calculated chord and twist of the current design

Aerofoils	r (m)	r/R	Chord (m)	Twist (degree)
Cylinder	0	0.000	7.61	0.00
	3.906	0.031	7.61	14.50
FFA-W3-600	15.498	0.123	7.72	14.43
	23.814	0.189	8.30	12.54
FFA-W3-480	29.484	0.234	8.64	9.96
FFA-W3-360	36.666	0.291	8.77	7.83
	43.218	0.343	8.62	6.72
FFA-W3-301	51.282	0.407	8.19	5.62
FFA-W3-241	71.19	0.565	6.64	2.89
	84.924	0.674	5.46	0.90
	101.93	0.809	4.10	-1.14
	119.95	0.952	2.69	-2.95
	126	1.000	1.19	-3.43

The blade shape has been compared with “NREL 5 MW RWT [11], DTU 10 MW RWT [12], and 10 MW WT [3]” as shown in Figure 4.7.1. This comparison has been made to ensure that the force that generates the torque will be distributed uniformly on the designed blade and to follow the similar blade profiles. The “DTU 10 MW RWT” and the current blade designs show a good agreement. The maximum chord has been increased from 6.2 *m* to 8.77 *m* but the twist angle is identical to “DTU 10 MW RWT”.

The twist angle adjustment method will result in a high maximum power coefficient C_{Pmax} and uniformly distributed aerodynamic forces throughout the blade, but it does not ensure optimal performance [3]. To obtain the optimal performance a variable pitch and variable rotational speed wind turbine blade must be designed which is included in the current PhD research.

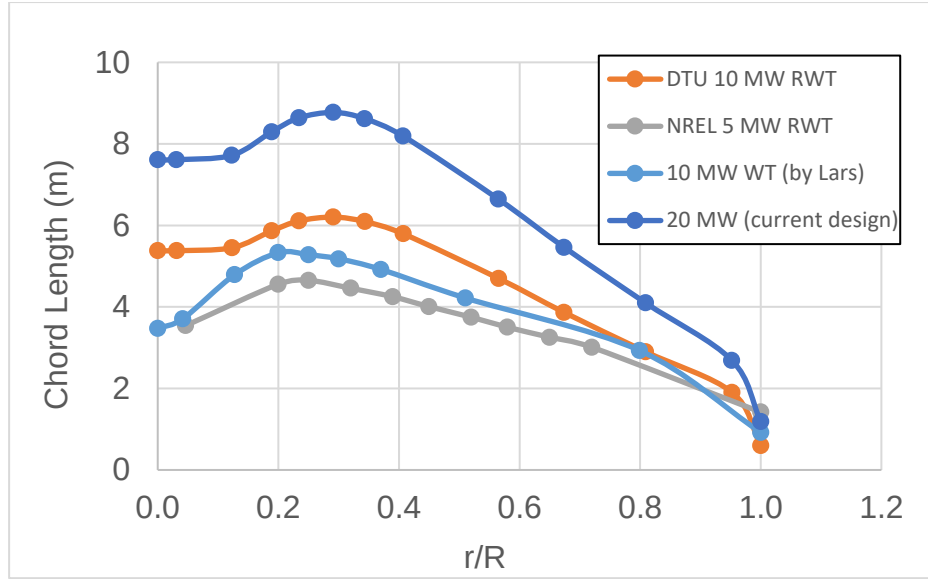


Figure 4.7.1: Comparison of chord length and shape of the blade

4.7.1 Upscaled Blade Geometry

Parameters from Table 4.7.1 have been imported into QBlade to generate blade geometry (Figure 4.7.2). The blade's thread axis in [% chord] are identical to those used for the length optimization (c.f. Table 4.4.2). All key parameters in Table 4.4.1 remain the same apart from the tip speed ratio and the rated wind speed.

The tip speed ratio has changed from 9 to 7.5 and the rated wind speed has changed from 12 m/s to 11 m/s . When the chord of the blade is increased the tip speed ratio will decrease which in turn reduces the noise. Because the noise is approximately proportional to the sixth power of the TSR. Increasing the chord will provide extra room for ensuring structural stability. Increasing the chord length will lead to improved power coefficient and thrust coefficient. Hence forth increasing the extraction of KE from the free stream wind. Increasing the length of the blade increases the power due to its large swept area. Whereas increasing the chord will increase the efficiency of the wind turbine through an improved power coefficient. These increments will lead to reduction in TSR.

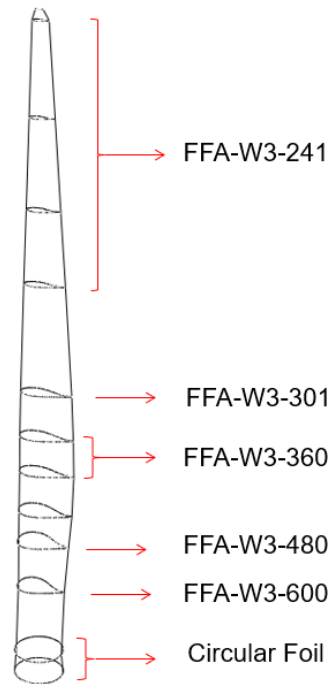


Figure 4.7.2: Blade geometry for chord and twist optimization

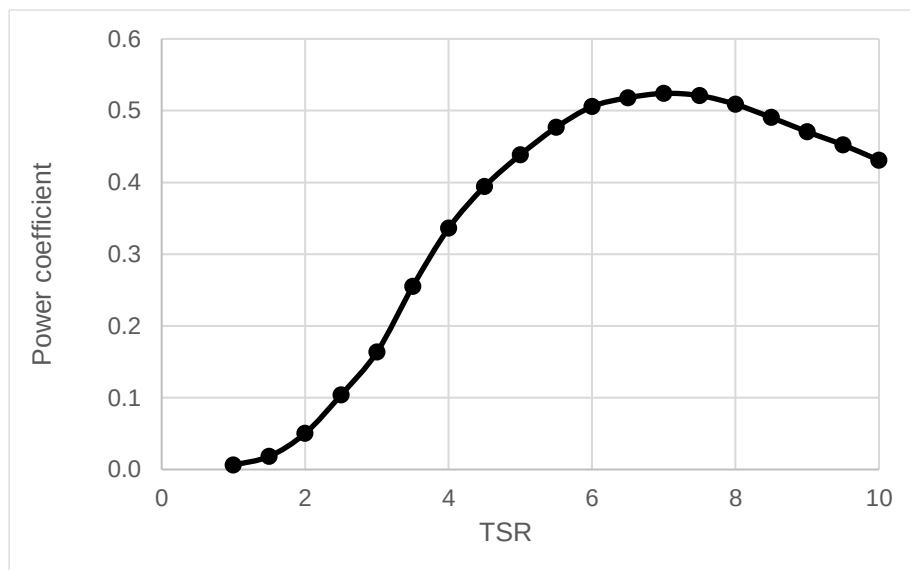


Figure 4.7.3: Power coefficient vs tip speed ratio for chord and twist optimization

The maximum C_p of 0.516 for the current blade was obtained at a tip speed ratio of 7.5 and a pitch angle of 0 *degree*, as shown in Figure 4.7.3. Note that

C_p vs TSR is a single curve for any shape or configuration of the blade, and it does not change with wind speed.

4.7.2 Results

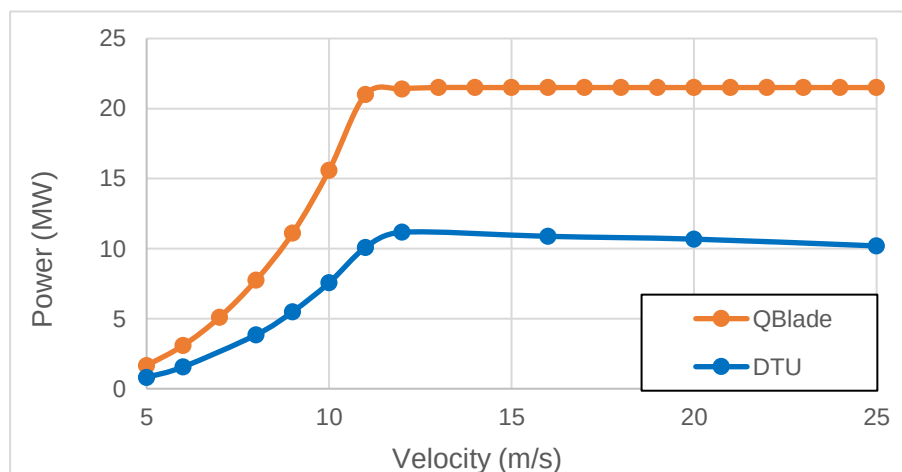
Similarly, to the previous design case, the current blade was also simulated for different rotational speeds and pitch angles. The simulation parameters used to calculate the power and thrust will also remain the same ($\rho = 1.225 \text{ kg/m}^3$, $\mu = 1.784 * 10^{-5} \text{ kg/(m.s)}$). The velocity ranges between 5 m/s and 25 m/s, and the rated power output was achieved at 11 m/s.

Optimized rotational speeds were selected in between 4 and 10 rpm with respect to the velocity, keeping the blade at zero pitch angle. To achieve 20 MW, the pitch angle of higher velocity was varied between 0 and 30 degree. The maximum obtained pitch angle was 21.92 degree. For the length optimization, the pitch angle was zero for velocities from 5 m/s to 11 m/s, but in here, the pitch angle is zero from 6 m/s to 11 m/s. The pitch angle at 5 m/s was optimized to be 1.22 degree to increase the power from 1.49 MW to 7.6 MW and to decrease the thrust from 0.835 MN to 0.76 MN. Note that the defined pitch angles are positive for the nose down. Table 4.7.2 provides the detailed aerodynamic results from this study, and the operational parameters based on QBlade for different velocities.

Those results have been compared with the “DTU 10 MW RWT” based results to establish whether the power curve follows the ideal curve. Figure 4.7.4 shows the power and thrust versus velocity curves. Both plots patterns match well, and the power has increased compared to the DTU-based one. Figure 4.7.5 shows the non-dimensional power coefficient and thrust coefficient plots. The maximum rated power of 21 MW was achieved at 11 m/s and was constant till the cut-out speed. Similarly, the maximum thrust of 3.42 MN was achieved at 11 m/s.

Table 4.7.2: Operational parameter and aerodynamic results at different velocities

Velocity (m/s)	Rotational speed (RPM)	Pitch angle (degree)	Power (MW)	C_p	Thrust (MN)	C_t
5	4.1	1.22	1.7	0.432	0.76	0.995
6	4.1	0	3.1	0.468	1.08	0.978
7	4.5	0	5.1	0.487	1.41	0.942
8	5	0	7.8	0.495	1.82	0.931
9	5.6	0	11.1	0.498	2.30	0.929
10	6.2	0	15.6	0.511	2.84	0.930
11	6.8	0	21.0	0.516	3.42	0.925
12	7.2	6	21.4	0.405	2.48	0.564
13	7.2	8	21.5	0.320	2.13	0.413
14	7.2	9.65	21.5	0.256	1.95	0.326
15	7.2	11.06	21.5	0.209	1.80	0.262
16	7.2	12.42	21.5	0.172	1.68	0.215
17	7.2	13.65	21.5	0.143	1.61	0.182
18	7.2	14.84	21.5	0.121	1.52	0.154
19	7.2	15.92	21.5	0.103	1.47	0.133
20	7.2	16.97	21.5	0.088	1.42	0.116
21	7.2	18.06	21.5	0.076	1.37	0.102
22	7.2	19.04	21.5	0.066	1.34	0.091
23	7.2	20.05	21.5	0.058	1.30	0.080
24	7.2	21.03	21.5	0.051	1.26	0.071
25	7.2	21.92	21.5	0.045	1.24	0.065



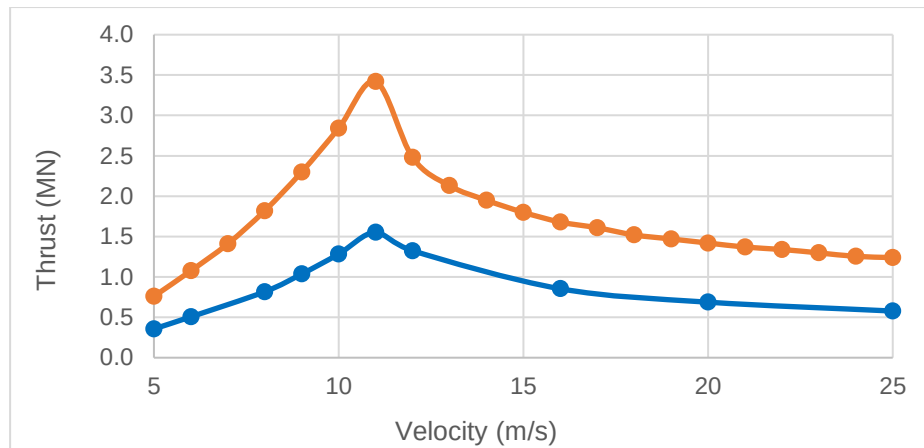
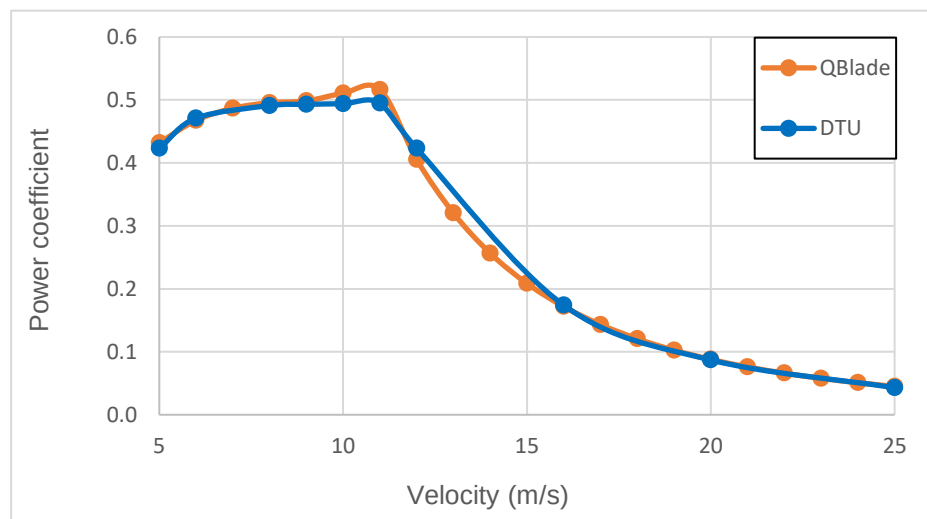


Figure 4.7.4: Power and thrust versus velocity comparison

The power coefficient and thrust coefficient obtained with QBlade agree well with the “DTU 10 MW RWT” results. The thrust coefficient has slightly increased below the rated velocity. However, after the rated velocity, the thrust matches well the baseline results. A CFD study has been carried out for a few velocities only, and results compared with those from QBlade and DTU, this is detailed in the next section



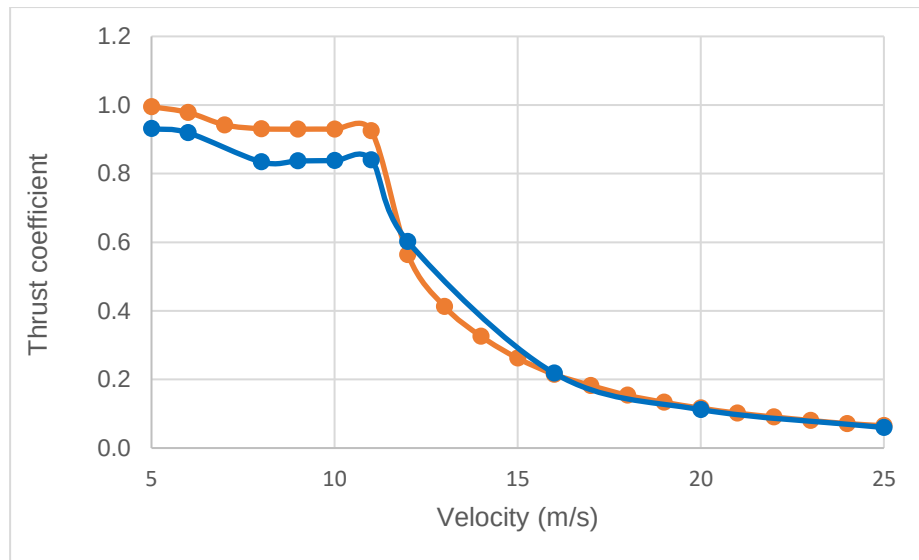


Figure 4.7.5: Power coefficient and thrust coefficient versus velocity comparison

4.8 Upscaled Blade Chord Length and Twist Angle Study Using CFD

All the geometrical parameters have been imported into ICEM CFD to build the optimized blade. The geometry was meshed, and the meshed file imported into FLUENT for the flow analysis and to calculate the aerodynamic results. Figure 4.8.1 shows the final optimized blade geometry in terms of length, chord, and twist.



Figure 4.8.1: Final blade geometry

4.8.1 Meshing

A hexa-mesh was created on the new blade design. The domain size was scaled up according to size of the blade. An increase of the size of the domain with an increase of the geometry size will avoid any flow interaction between the domain boundaries and the geometry. As the flow is a rotational flow, the domains must be large enough so that the flow leaves the domain outlet without any flow reversal, i.e., the flow must reach atmospheric pressure before it reaches the outlet. Figure 4.8.2 shows the mesh on the blade and at cut sections.

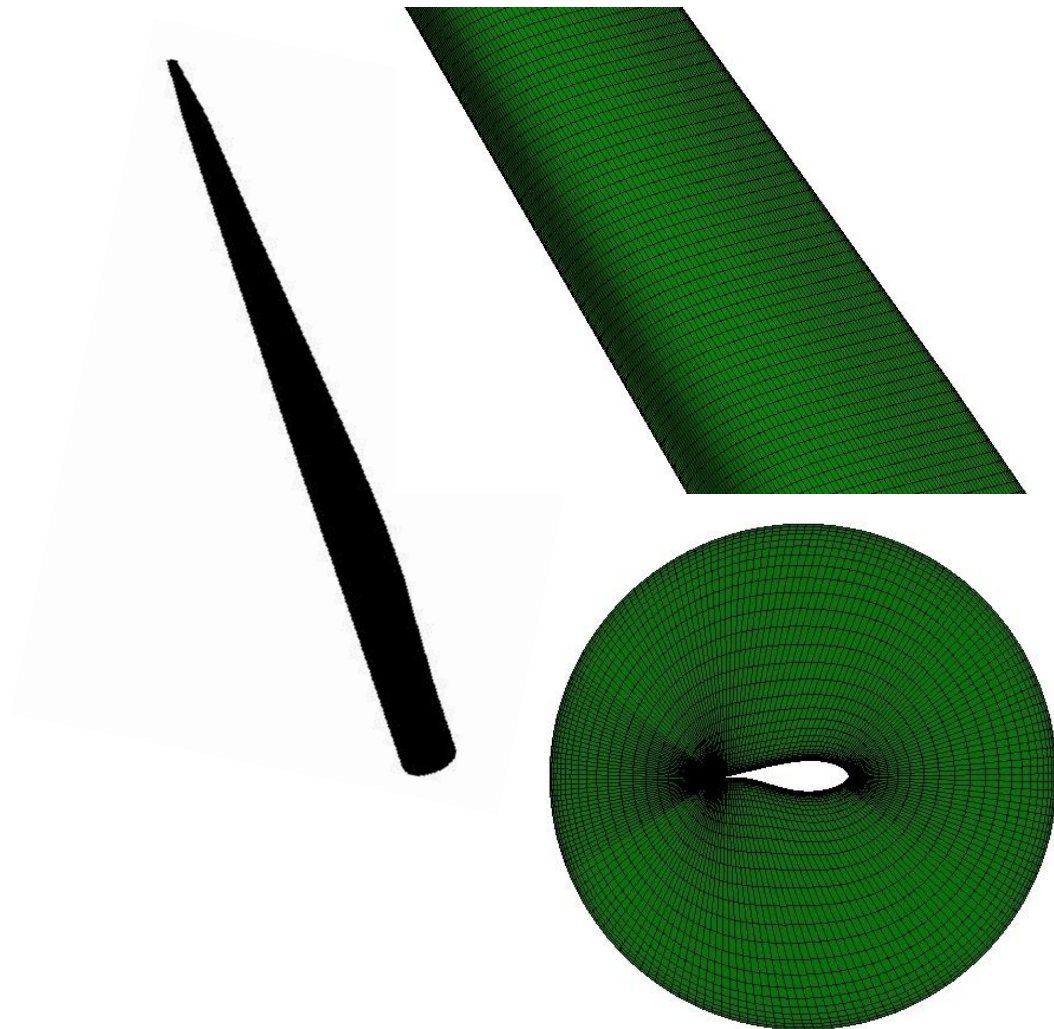


Figure 4.8.2: Mesh on the blade, at a section, and at a cross section

4.8.2 Results

The meshes are imported in FLUENT, and the same boundary conditions as for previous cases are implemented. The rotational speed and pitch angle are changed according to the current QBlade aerodynamic performance parameters. The pitched blades did not give expected results because of the pitch angle sensitivity. Hence, a pitch angle optimization has been carried out for a pitched blade at higher velocities. Table 4.8.1 shows the reduction in pitch angles studied.

Table 4.8.1: Pitch angle sensitivity study

Velocity (m/s)	Original Pitch angle (degree)	Reduced pitch angle by degree	Optimized pitch angle by degree	Optimized pitch angle (degree)
5	1.22	0, 1	0	1.22
12	6.00	1	1	5.00
16	12.42	1, 1.25	1.25	11.17
20	16.97	1.25, 1.35, 1.4, 1.5	1.5	15.47
25	21.92	1.35, 1.5, 1.65	1.65	20.27

The pitch angle is sensitive at high velocities, hence, the pitch angle at 5 m/s increased the thrust by 8% and the power was reduced by 8% when the pitch angle was reduced by 1 *degree*. Increasing the thrust will increase the bending of the blade, which will then hit the tower and damage the blade and tower. Decreasing the power by 8% will lead to overall losses in the long term. Hence the pitch angle for 5 m/s was not changed to obtain the maximum power output.

For higher velocities, from 12 m/s to 25 m/s the pitch angle was reduced from the optimum pitch angle reduction obtained in the baseline study as explained previously for the length optimization. In this section, pitch angle sensitivity studies for 20 m/s and 25 m/s will be detailed. The results of the remaining pitch angles are reported in the appendix section 7.4.

Table 4.8.2 Pitch angle optimization carried out for 20 m/s

	Pitch angle sensitivity study (CFD)		QBlade results		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
15.72	1.27	19.37	1.42	21.5	-10.62	-9.90
15.62	1.31	20.06			-8.09	-6.69
15.57	1.32	20.40			-6.71	-5.11
15.47	1.37	21.17			-3.80	-1.53

For velocity 20 m/s, four pitch angles have been considered to obtain the optimized aerodynamic results. The optimized pitch angle was selected, based on the percentage of error. A minimum error of -3.80% and -1.53% were obtained for the thrust and power, respectively, at a pitch angle of 15.47 degree. Further reduction was not carried out as the obtained errors were deemed acceptable. The blade was designed to achieve 20 MW but from the current design and aerodynamic performance parameter, the expected power has been increased by 5.85%. This increment would make sure that the expected power of 20 MW will be achieved even if there is a reduction in power with respect to structural or manufacturing defects. Wind turbines tend to reduce the power output down the line by 5% to 20 % due to erosion on the leading edge and corrosion occurring in a wind farm. Such erosion could be due to rough weather at sea, droplets/pellets impact due to heavy rain conditions, and corrosion due to salted water from the sea leading to cracks [124].

Figures 4.8.3 and 4.8.4 show the pressure coefficient C_p and the pressure contour plots for 20 m/s at different pitch angles. As previously, the plots are generated at 30% and 60% of the span of the blade. Thrust and power were increased by 6.82% and 8.37%, respectively, when applying a pitch angle from 15.72 degree to 15.47 degree.

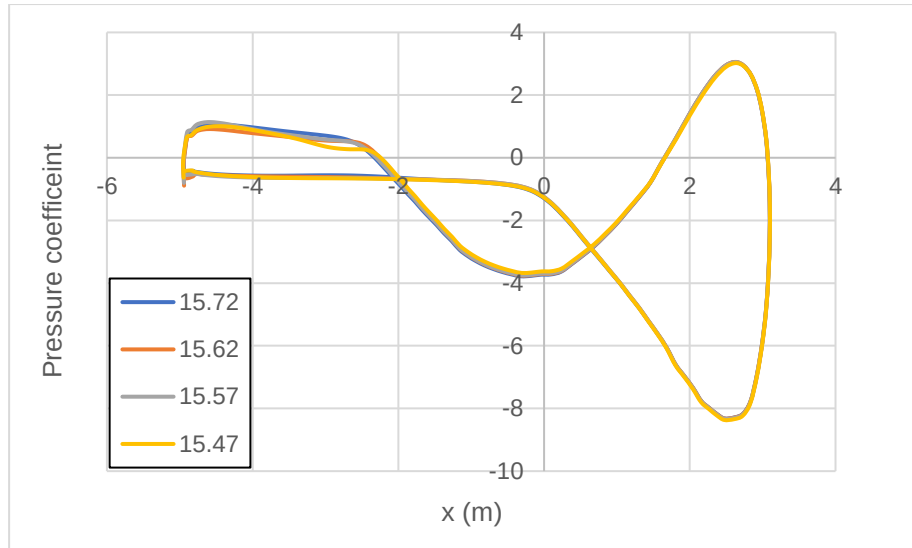
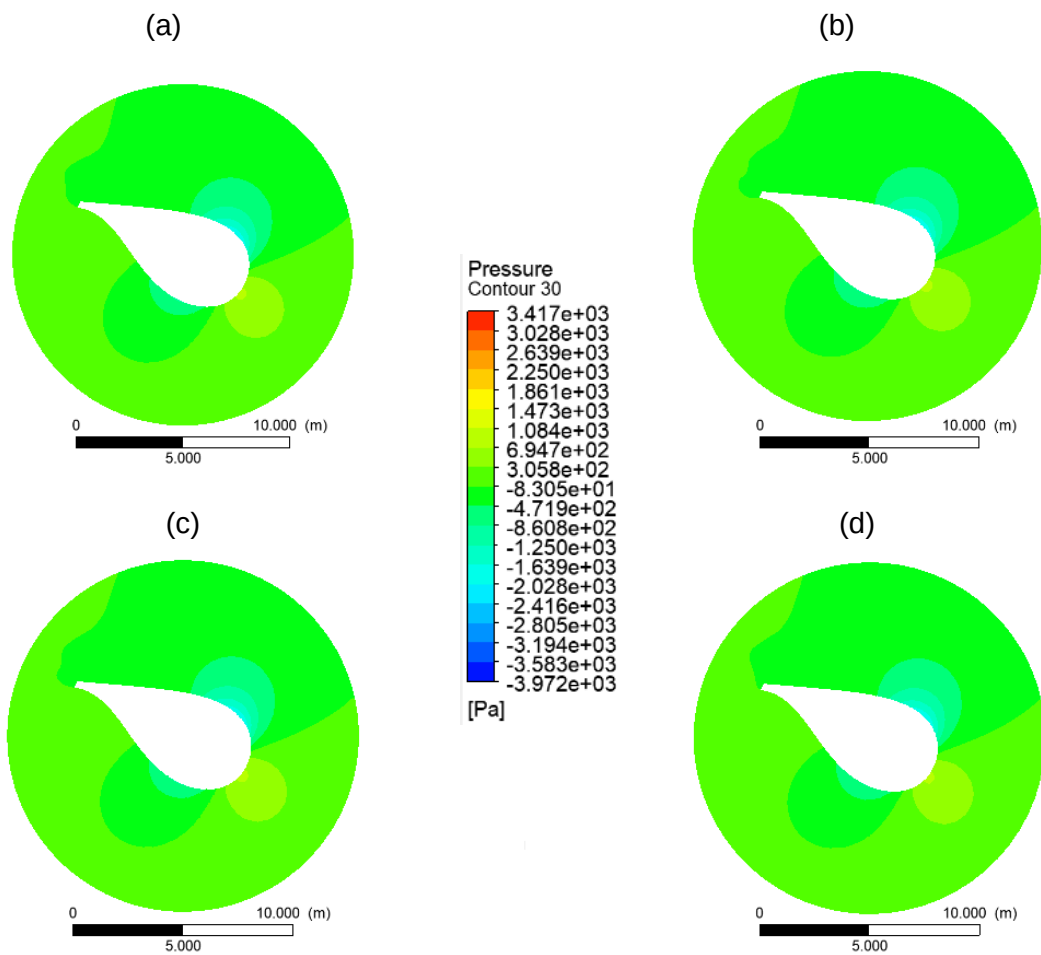


Figure 4.8.3: Pressure coefficient C_p and pressure contour plots for velocity 20 m/s for different pitch angle (a) 15.72 degree (b) 15.62 degree (c) 15.57 degree (d) 15.47 degree at span 30%

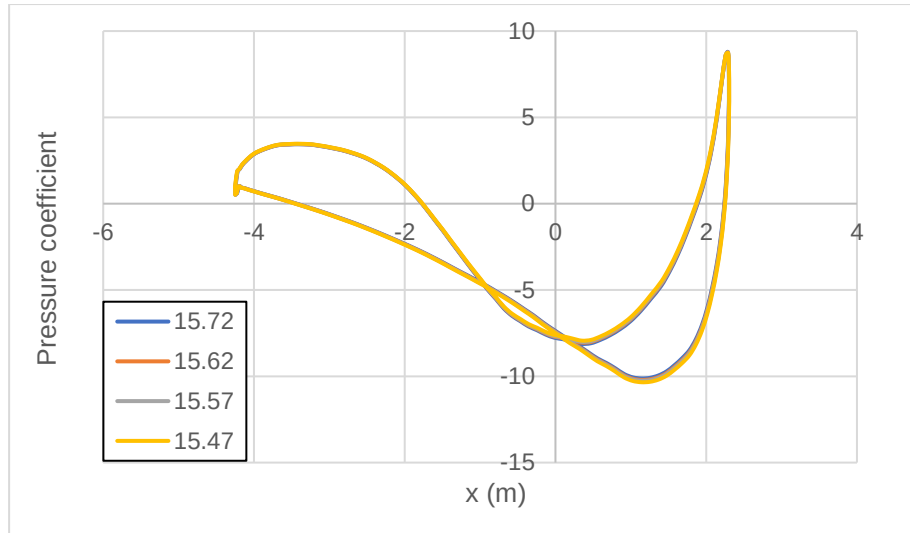
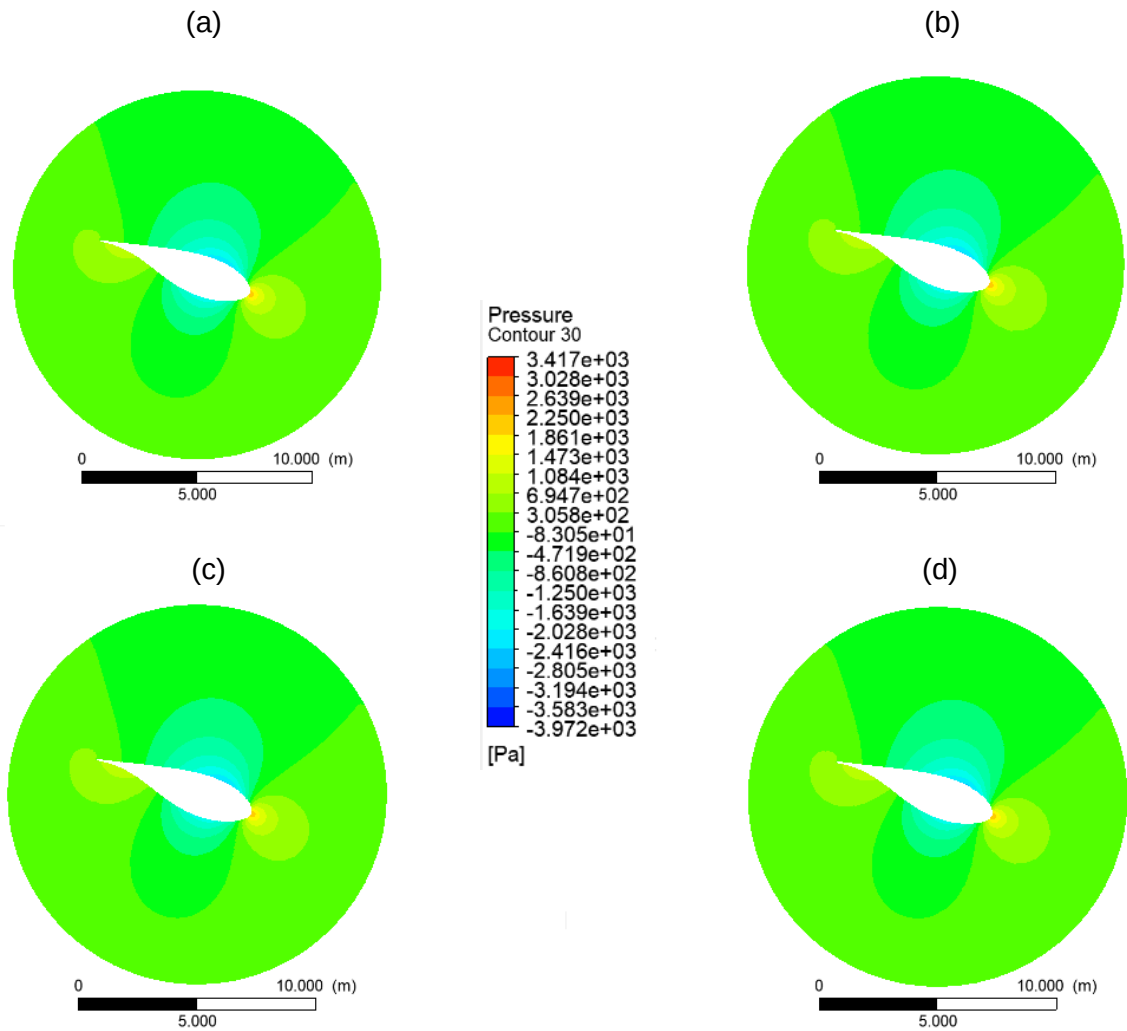


Figure 4.8.4: Pressure coefficient C_p and pressure contour plots for velocity 20 m/s for different pitch angle (a) 15.72 degree (b) 15.62 degree (c) 15.57 degree (d) 15.47 degree at span 60%

Table 4.8.3 Pitch angle optimization carried out for 25 *m/s*

	Pitch angle sensitivity study (CFD)		QBlade results		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
20.57	1.03	18.01	1.24	21.5	-16.54	-16.22
20.42	1.09	19.54			-11.70	-9.14
20.27	1.15	20.89			-7.00	-2.84

Similarly, studies have been carried out for 25 *m/s*. Table 4.8.3 shows the pitch angle optimization for this velocity. The required power output was achieved at a pitch angle of 20.27 *degree* with an error of -7% and -2.84% for the thrust and power, respectively. Further reduction in pitch angle was not carried out as the expected power was increased by 4.45 %. All the pitch angles were reduced to achieve 20 MW of power output in the CFD study, but when designing the blade geometry in QBlade, the aerodynamic performance parameters were selected in such a way that the power was 5% to 7.5% higher than the required power. This was performed to avoid redoing a domain and boundary condition sensitivity study. When doing the pitch angle sensitivity analysis, the pitch angle was reduced such that the power output reached the desired power output.

Figures 4.8.5 and 4.8.6 show the pressure coefficient C_p and the pressure contour plots for 25 *m/s* and different pitch angles at 30% and 60% of the blade span. The thrust and power were increased by 9.54% and 13.38%, respectively, and the pitch angle changed from 20.57 *degree* to 20.27 *degree*.

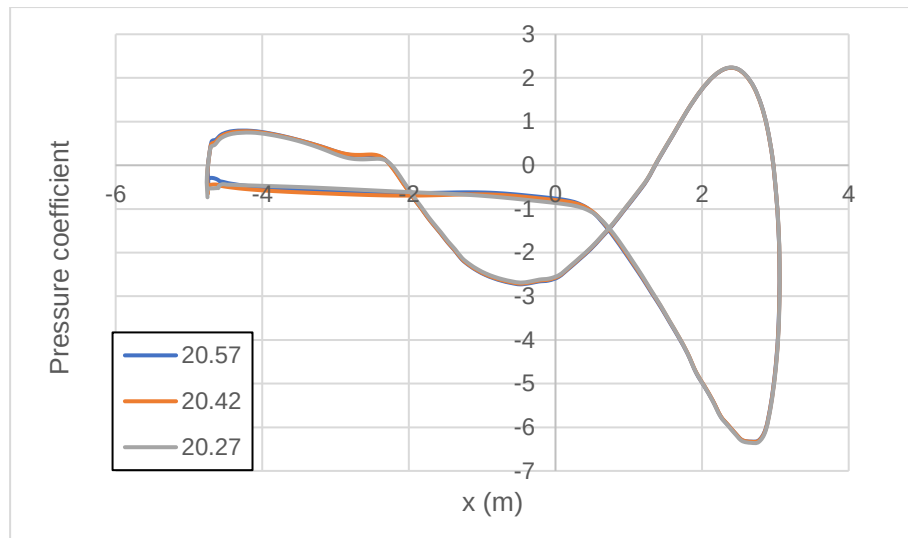
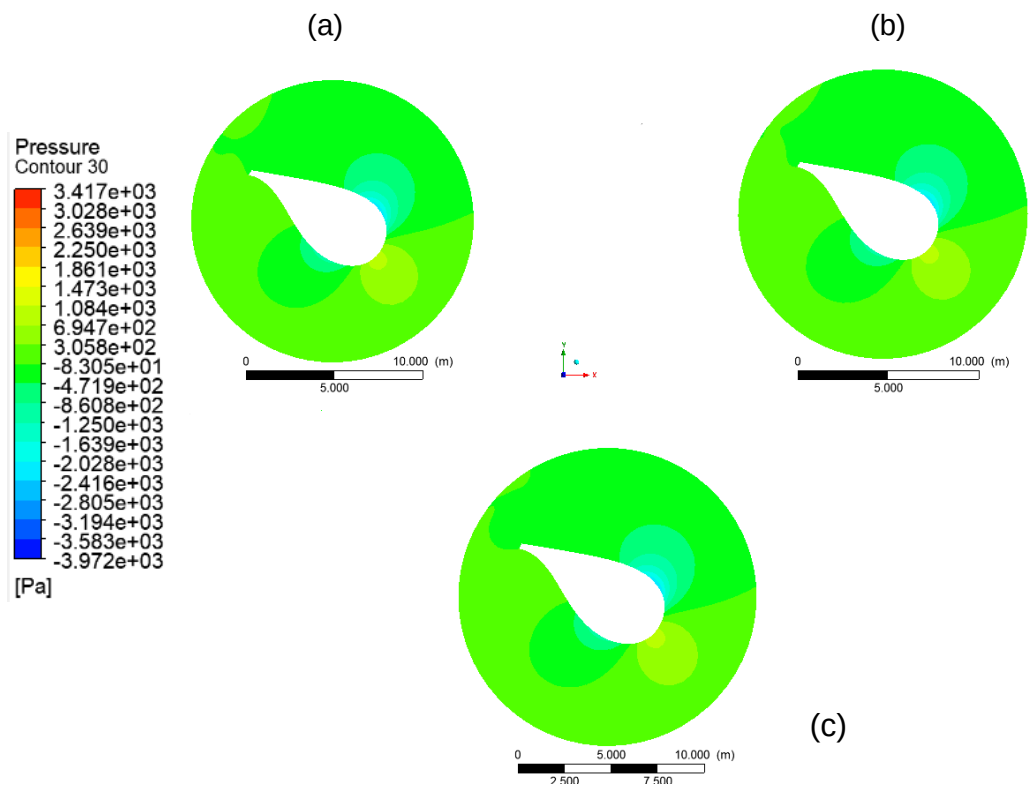


Figure 4.8.5: Pressure coefficient C_p and pressure contour plots for velocity 25 m/s for different pitch angle (a) 20.57 degree (b) 20.42 degree (c) 20.27 degree at span 30%

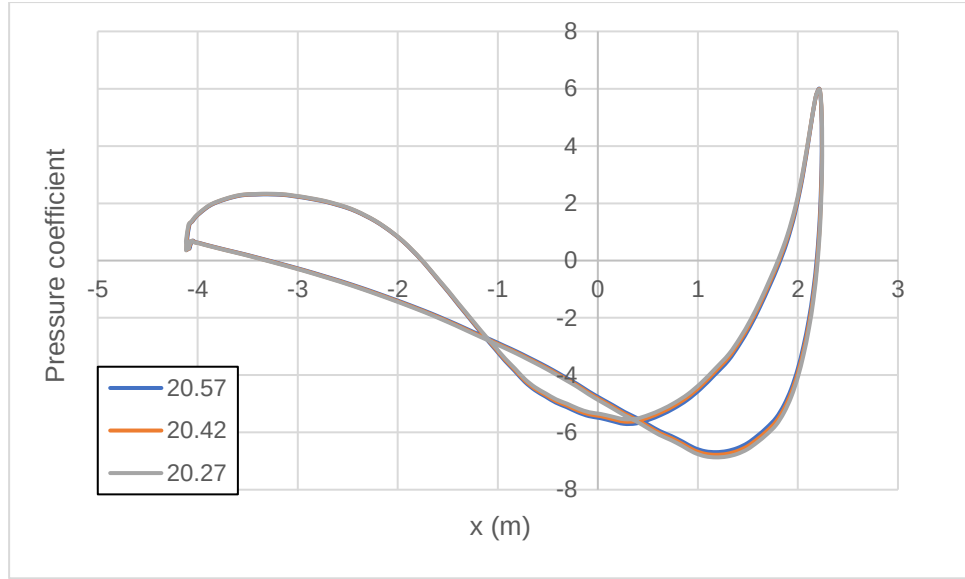
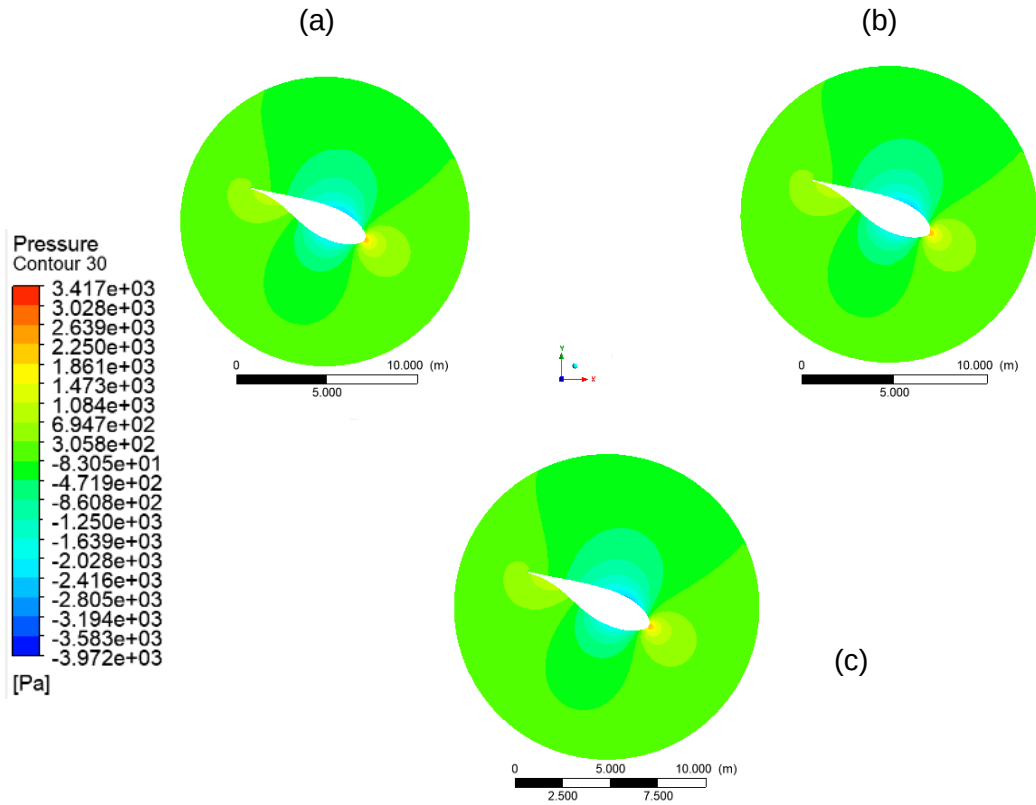
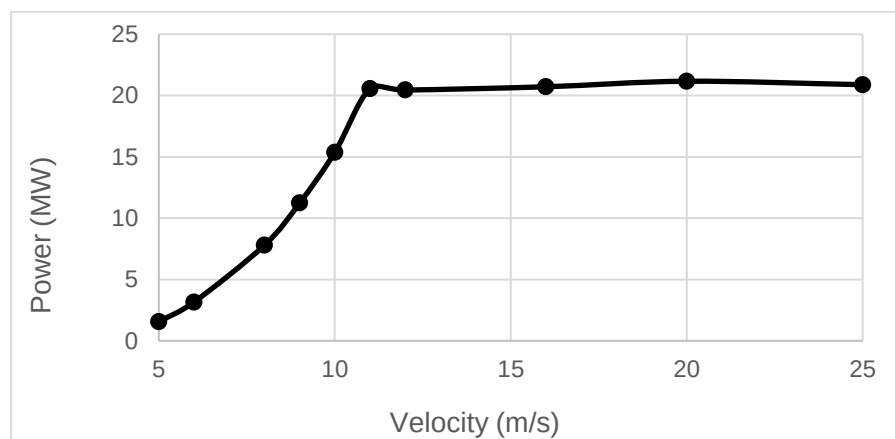
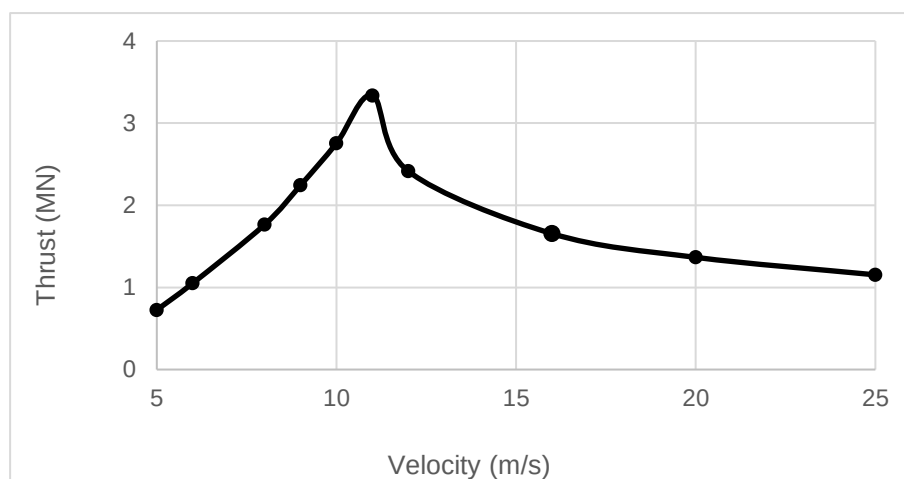


Figure 4.8.6: Pressure coefficient C_p and pressure contour plots for velocity 25 m/s for different pitch angle (a) 20.57 degree (b) 20.42 degree (c) 20.27 degree at span 60%

Table 4.8.4 and Figure 4.8.7 show the chord and twist optimization results.

Table 4.8.4: Aerodynamic results from CFD for chord length and twist angle optimization

Velocity (m/s)	Rotational speed (RPM)	Pitch angle (degree)	Thrust (MN)	C_t	Power (MW)	C_P
5	4.1	1.22	0.73	0.95	1.563	0.409
6	4.1	0	1.05	0.96	3.157	0.478
8	5	0	1.76	0.90	7.794	0.498
9	5.6	0	2.24	0.91	11.235	0.504
10	6.2	0	2.75	0.90	15.366	0.503
11	6.8	0	3.34	0.90	20.572	0.506
12	7.2	5	2.41	0.55	20.455	0.387
16	7.2	11.17	1.66	0.21	20.719	0.166
20	7.2	15.47	1.37	0.11	21.401	0.083
25	7.2	20.27	1.15	0.06	20.889	0.044



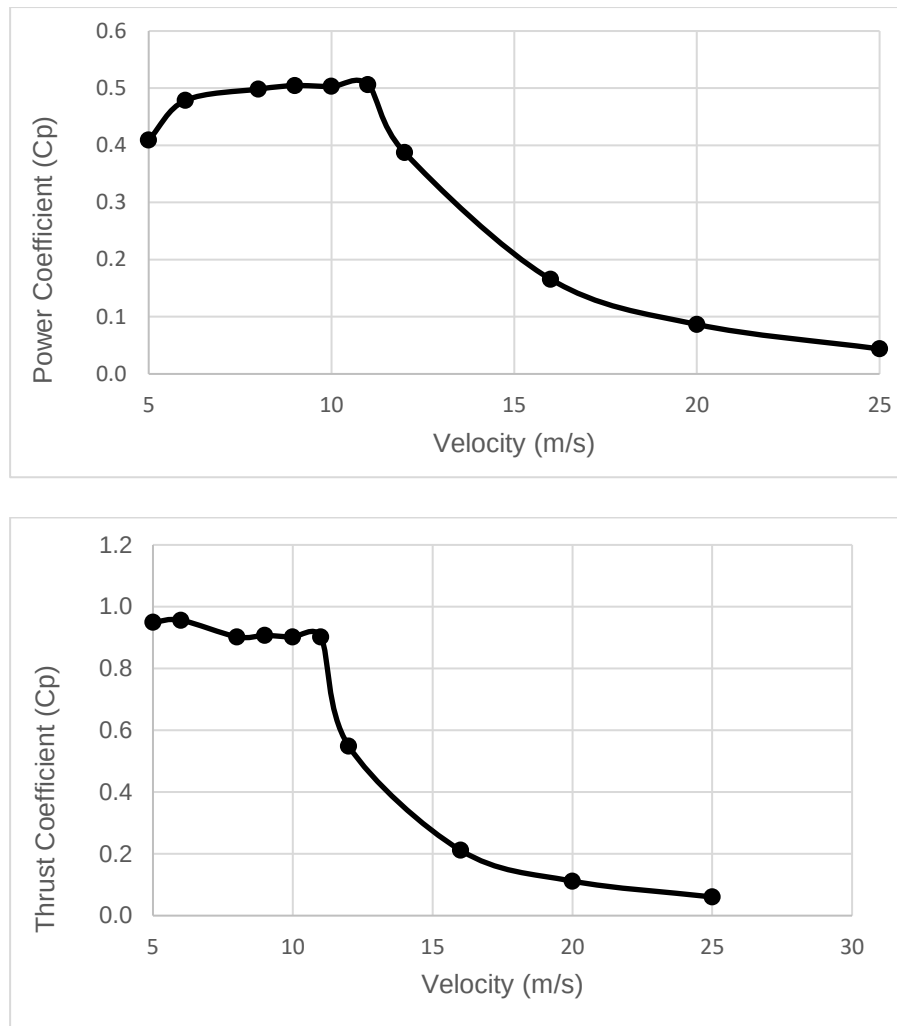


Figure 4.8.7: Chord and twist optimization results

4.8.3 Discussion

For the current case study, the new chord and twist were established for a 126 m long blade. The calculations were based on the blade element theory, the momentum theory and Betz' equations. Few assumptions were made to simplify chord Equation 4.17. From the length optimization study, although the required power of 20 MW was already achieved, both the chord and twist were optimized also; this was performed as the blade will be facing bending effects due the pressure on its surface. Such continuous bending will lead to fatigue and potential structural damage before the expected lifespan.

This problem can however be resolved with a better structural stability. Increasing the chord will give sufficient space for the structural design to resist

the bending moment. The second reason was to increase the thrust and power at low velocity i.e., below the rated velocity.

A new blade design was generated from the calculated chord and twist in QBlade. The aerodynamic performance parameters were selected for this blade, the design has a variable rotational speed and a variable pitch angle. The QBlade results were then compared with the “DTU 10 MW RWT” based ones to check if the results follow the ideal curve pattern. Compared to the “DTU 10 MW RWT” based results, the thrust was increased by 119.9 % and the power was increased by 108.2 % at a rated velocity of 11 m/s . Similarly, the thrust was increased by 17.1 % and the power was increased by 10.5 % at a rated velocity of 11 m/s when compared to the results obtained in the length optimization (c.f. Table 4.5.4). To check those initial results, CFD simulations were performed.

The blade was redesigned and meshed and exported to the flow solver. Similarly, to the baseline and length optimization, a pitch angle sensitivity study was carried out at higher velocities. From this sensitivity study, the power and thrust were increased by 103.92 % and 114.56 %, respectively, compared to the baseline “DTU 10 MW RWT” case.

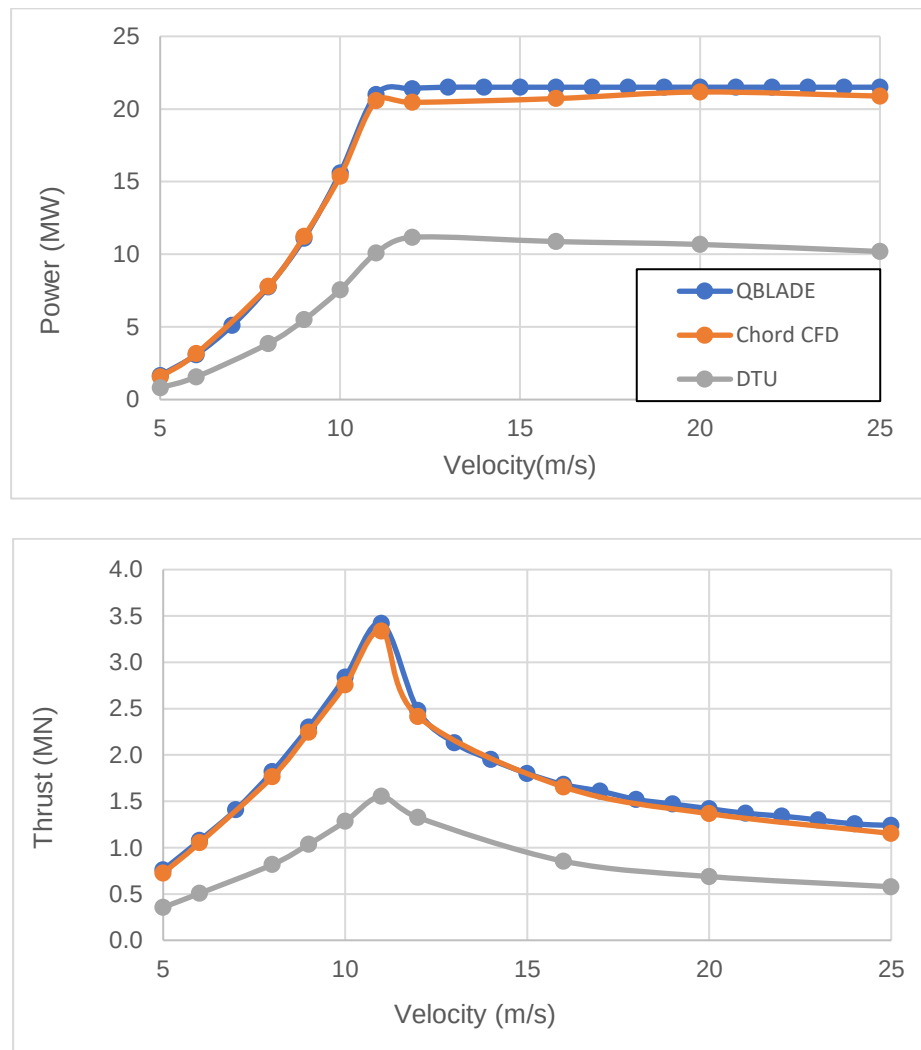
Table 4.8.5: Percentage of error between QBlade and CFD results

C_t	C_p
-4.56	-5.27
-2.26	2.33
-3.05	0.57
-2.43	1.21
-3.00	-1.50
-2.44	-2.04
-2.63	-4.41
-1.49	-3.63
-3.80	-1.53
-7.00	-2.84

Table 4.8.5 shows the percentage of error obtained in terms of thrust coefficient and power coefficient between QBlade and CFD. A maximum error of -7% and -5.27% was obtained for the thrust and power coefficients, respectively. Such percentage of error were deemed acceptable because the

differences are inherent to turbulence and viscosity effects, which are not accounted for in QBlade. The overall study shows that the final blade design and the aerodynamic performance parameters will produce 20 MW power output.

Figure 4.8.8 shows the aerodynamic results comparison between the baseline (DTU), QBlade and the CFD studies. These results follow the ideal curve pattern, and QBlade and CFD results agrees well with each other.



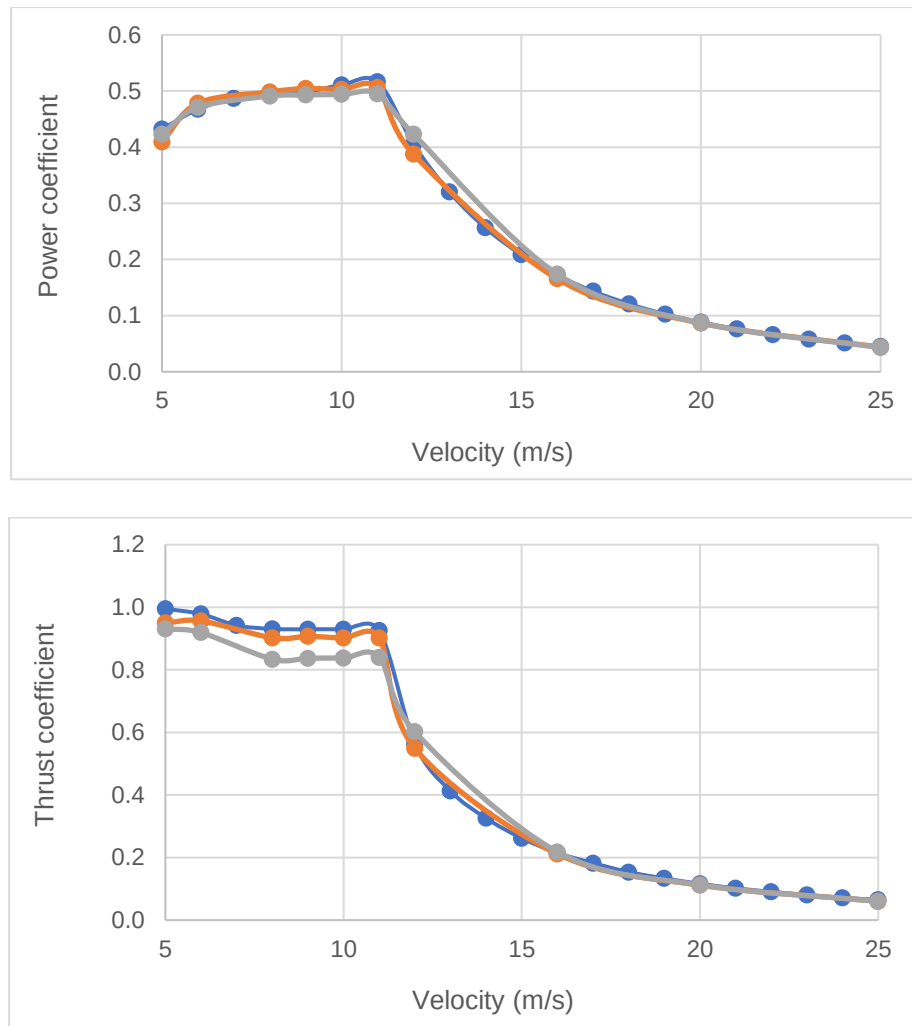


Figure 4.8.8: Aerodynamic results comparison of chord length and twist angle optimization

4.9 Comparison

The design variants in the current research i.e., length, chord and twist optimized design were studied for the aerodynamic performance parameters including variable rotational speed and pitch angles. The variation in thrust and power was compared with respect to velocities which is illustrated in Figure 4.9.1. The results are compared between the baseline case (“DTU 10 MW RWT”), the length optimized, and the chord-twist optimized design. Compared to the baseline, the thrust and power were increased in the new design variants. From the length optimization and the chord-twist optimization, the thrust was improved below the rated velocity of 11 m/s . The power of both design variants

remained almost the same even if there is a slight increase in overall power in the final optimized design.

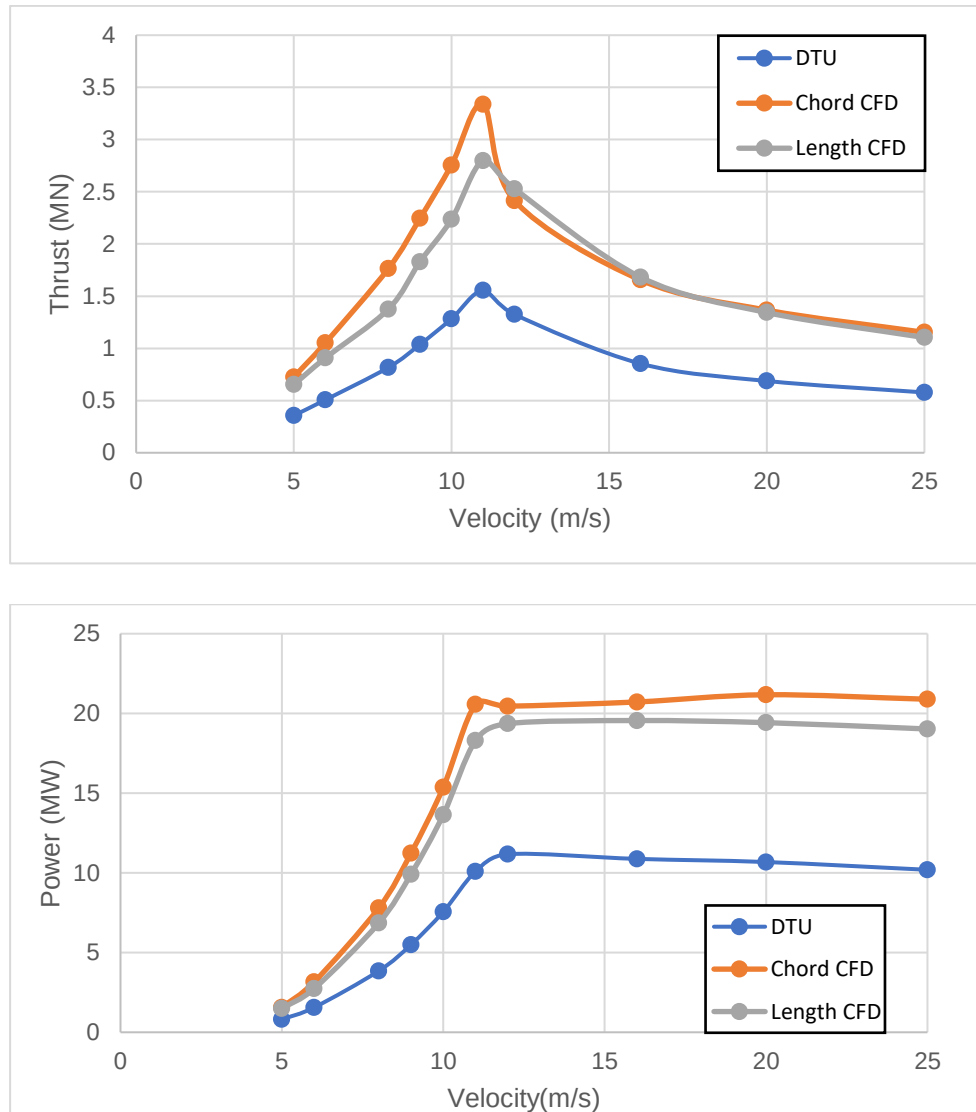


Figure 4.9.1: Thrust and power versus velocity curve for design variants

The validation is an important stage of the research to prove that the research carried-out is acceptable and accurate. Usually, numerical results are validated against experimental data. However, in the current research, due to unavailability of experimental results (impossible to generate for such a large design), a verification has been done by comparing the obtained power curve from the final wind turbine blade design with different lower power generating wind turbines. Such wind turbine results are widely accepted, are published in the literature, and have been used as baseline for various studies [12][11][125]. SWT-6.0-154 is an offshore wind turbine with rotor diameter of 154 *m* and rotor

speed between 5 to 11 rpm. This wind turbine's manufactured by Siemens [126]. The published power curve of SWT-6.0-154 is used to validate the current power curve.

Compared to all the wind turbine results shown in Figure 4.9.2, the power curve of the current design agrees with the pattern, hence, this design most certainly describes a valid 20 MW offshore wind turbine blade design.

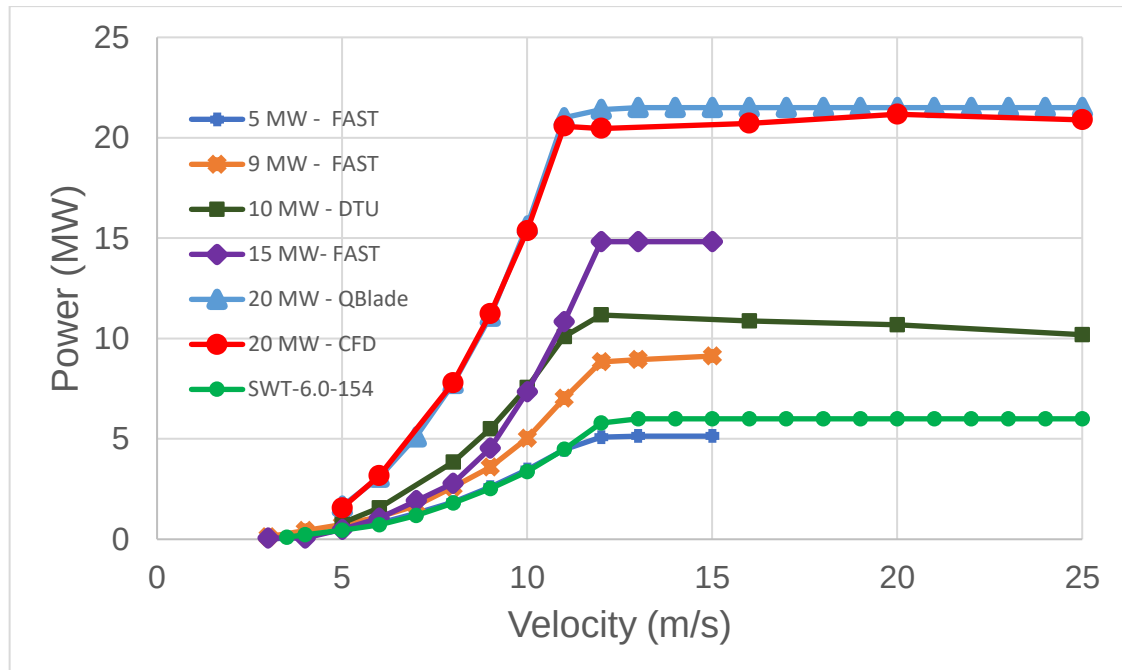


Figure 4.9.2: Power curve comparison of different wind turbine models.

5. Conclusions and Future work

5.1 Conclusions

The research presented in this thesis, a 20 *MW* offshore wind turbine has been designed and the aerodynamic performance has been outlined. A thorough literature review has been carried out on large scale offshore wind turbines. The existing current largest off-shore wind turbine is the Haliade-X (GE) which has set a trademark at producing 14 *MW*, 13 *MW* or 12 *MW* with a rotor diameter of 220 *m*. Vesta is also expecting to install a V236-15 *MW* prototype wind turbine in 2022. Based on the literature survey and existing information, a research gap has been identified for this PhD.

Increasing the annual energy production (AEP) has been the main motive for this research. The annual energy production can be improved by two different methods. Firstly, an upgrade of the existing wind turbines could be carried out by installing vortex generator, winglets or by waxing blade surfaces to remove the erosion at the leading edge. Secondly a new generation of wind turbines could be designed and installed, which would produce higher power. Hence in this study, the second approach has been considered, and a 20 *MW* offshore wind turbine has been designed with a rotor diameter of 252 *m*.

The baseline model has been studied for boundary condition (BC) interactions and BC placement sensitivity. This study has highlighted that a cylindrical domain with one blade only forms a more efficient model, due to a significant gain in simulation time and resources when compared to the whole 3-bladed model. Simulations for an inlet boundary placed 3D and 6D upstream of the wind turbine have been carried out, and the 3D upstream case showed satisfactory results. This model has thus been selected to reduce further computing time and resources. The outlet has been tested at 7D, 10D, 15D, and 20D downstream positions from turbine. Results, flow interactions, and turbulence intensity have been shown to work well when considering the outlet located 15D far from the turbine. The top surface of the domain has been set as a symmetry condition (4D high). A rotating cylindrical subdomain was designed at a radius of 1.5R (radius) around the blade. Pressure based steady-

state and transient simulations were carried out with different viscous models including laminar, Spalart-Allmaras, k-epsilon (standard and realizable), k-omega SST, transition k-kl-omega, and DES. The k-omega SST turbulence model was finally selected for the research.

The rotor blade was rotated to a given pitch angle at high velocity, between 12 *m/s* and 25 *m/s*. A pitch angle sensitivity study was carried out to obtain accurate results, which were then compared against both QBlade and EllipSys3D based results.

Using similarity rules, the blade length was modified to achieve a power of 20 *MW*. The new design was meshed, and the simulations were carried out under design and off-design conditions. As the blade length was increased, the rotational speed and pitch angle of the blade were also changed. Hence, simulations were carried out for different rotational speeds and pitch angles, and the optimum rotational speed and pitch angle were selected for further design variants.

These results were compared with the baseline results and the power and thrust was increased by 80.84% and 88.67% at a rated wind velocity of 12 *m/s*. Although promising, this numerical study could not be finalised as the chord length along the span had to be increased to provide a better structural stability and to increase the thrust.

2D simulations were carried out on FFA-W3-241, FFA-W3-301, FFA-W3-360, and FFA-W3-480 aerofoils located along the blade length of the new optimised design. The corresponding CFD results were compared with QBlade based results. The angle of attack α at which C_D/C_L is minimum was selected. This angle of attack was used to calculate the chord length and the twist of the aerofoils.

To get an optimum rotor, a tip speed ratio $\lambda = 7.5$ was determined, this value corresponding to the maximum C_p achieved. Aerofoils along the span of the blade provided the normalized radius. From these normalized values, the angle of relative velocity, chord length and twist angles were calculated,

The calculated chord length and twist angle of the aerofoils were used to generate a new blade design and CFD simulation were performed. For the new design, simulations were carried out by varying the rotational speed and the pitch angle. In this case, the power and thrust generated were increased by 103.92 % and 114.56 %, respectively, compared to the baseline case.

The results of the final design variant were compared with 5 MW, 9 MW, 10 MW and 15 MW offshore wind turbines in terms of power and wind speed, to obtain a numerical check of the results, as field or experimental data are not available yet for such a large turbine.

The important novel aspects of the research carried out in this PhD project can be summarised as follows:

- **Model settings including boundary conditions (BC):** The BC sensitivity study was a major part of the research. BC interactions, BC placement sensitivity analysis, BC type, model, and solver settings had been defined well to obtain accurate results. These parameters were analysed, and a perfect setup was achieved. This setup can be used in any CFD study for small-scale or large-scale offshore wind turbines.
- **Pitch angle sensitivity study:** This study is another major part of any large-scale wind turbine blade aerodynamic performance. The simple blade element momentum theory's pitch angle is not as reliable as CFD, due to viscosity and turbulence effects not accounted for. It has been shown that the pitch angle must be reduced between 1 and 1.65 *degree*, depending on velocity. The current research proved that the velocities in the range of 12 *m/s* to 16 *m/s* need a pitch angle reduction between 1 and 1.25 *degree* whereas velocities ranging from 20 *m/s* to 25 *m/s* need a pitch angle reduction from 1.5 to 1.65 *degree* to achieve results close to expectations.

- **Length optimization:** Using similarity rules, the blade length was modified to achieve a power of 20 MW. It has been shown that the wind power of a blade mainly depends on the blade length.
- **Rotational speed and Pitch optimization:** Different rotational speed simulations were carried out for each velocity, and optimum rotational speed and pitch angle values were identified. If any of the aerodynamic parameters is varied, then the rotational speed and pitch angle of the rotor blade will vary.
- **Chord to thickness ratio optimization:** From Betz' optimum rotor, a new chord length was established. The thickness of the blade increases as the chord length increases in the ratio of 1:1.
- **Twist optimization:** The twist angle of the blade must decrease closer to the blade tip. Since the blade tip has a thin aerofoil, the L/D ratio is higher than at the base. Optimized chord length, thickness and twist angle values were quantified, and a new design was obtained. CFD simulations were carried out to establish the difference in power and thrust.

5.2 Recommendations for Future Work

The focus of this research was on the aerodynamic design optimization of a large-scale offshore wind turbine. Further studies should be carried out on the structural design optimization and the aero-servo-elastic design optimization aspects. The blades investigated in this work were assumed to be rigid, the study should be extended to blades with a prebend and flexible structure.

Although optimized high performing aerofoils were selected to obtain a 20 MW wind turbine in the current research, experiments might show differences when using different aerofoils, and the aerodynamic performance parameters might be affected. Although impractical for large turbines, experiments should be performed on small scale wind turbines using the current

designed blades, and numerical results should be compared to experimental data.

To improve further the performance of the blades and wind turbines, and to improve the life span of the blade, vortex generators could be installed. Winglets could also be installed to this blade; this would result in additional power output.

Finally, additional work could be performed to understand how efficient the designed blades operate under the harsh conditions encountered offshore. Wind turbines might face strong winds, ice formation, erosion, etc. This could be studied numerically and experimentally.

6. References

1. Wilson AB. Offshore Wind Energy in Europe. Brussels, Belgium. 2020; (November): 1–6. Available at: [https://www.europarl.europa.eu/RegData/etudes/BRIE/2020/659313/EPRS_BRI\(2020\)659313_EN.pdf](https://www.europarl.europa.eu/RegData/etudes/BRIE/2020/659313/EPRS_BRI(2020)659313_EN.pdf)
2. Mourad MG., Shahin I., Ayad SS., Abdellatif OE., Mekhail TA. Effect of Winglet Geometry on Horizontal Axis Wind Turbine Performance. Engineering Reports. 2020; 2(1): 1–19. Available at: DOI:10.1002/eng2.12101
3. Frøyd L., Dahlhaug OG. Rotor Design for a 10 MW Offshore Wind Turbine. Proceedings of the International Offshore and Polar Engineering Conference. 2011; 8: 327–334.
4. Thé J., Yu H. A Critical Review on the Simulations of Wind Turbine Aerodynamics Focusing on Hybrid RANS-LES Methods. Energy. 2017; 138: 257–289. Available at: DOI:10.1016/j.energy.2017.07.028
5. Ingram G. Wind Turbine Blade Analysis using the Blade Element Momentum Method. October. 2011; 1.1: 1–21. Available at: https://community.dur.ac.uk/g.i.ingram/download/wind_turbine_design.pdf
6. Mekhail TA., Fadeel WA., Elmagid WMA. Experimental and CFD of Designed Small Wind Turbine. International Journal of Science and Engineering Research. 2015; 6(1): 59–68. Available at: <https://pdfs.semanticscholar.org/74d6/04944416899058ed0df3c1fc2a224d7c0f42.pdf>
7. Johnson DA., Gu M., Gaunt B. Wind Turbine Performance in Controlled Conditions: BEM Modeling and Comparison with Experimental Results. International Journal of Rotating Machinery. 2016; : 1–12. Available at: <http://www.tandfonline.com/doi/abs/10.1080/15435075.2012.675799>
8. Akay B., Ragni D., Ferreira CS., Bussel GJW Van. Effect of Acceleration on Dynamic Stall of Airfoil in Unsteady Operating Conditions. Wind Energy. 2014; 19: 17–33. Available at: DOI:10.1002/we.1818

9. Grasso F. Usage of Numerical Optimization in Wind Turbine Airfoil Design. *Journal of Aircraft*. 2011; 48(1): 248–255. Available at: DOI:10.2514/1.C031089
10. Bjorck A. Coordinates and Calculations for the FFA-WI-xxx, FFA-W2-xxx, FFA-W2-xxx and FFA-W3-xxx Series of Airfoils for Horizontal Axis Wind Turbines. FFA TN1990-15. 1990.
11. Jonkman J., Butterfield S., Musial W., Scott G. Definition of a 5-MW Reference Wind Turbine for Offshore System Development. National Renewable Energy Laboratory. 2009. Available at: http://tethys-development.pnnl.gov/sites/default/files/publications/Jonkman_et_al_2009.pdf
12. Bak C., Zahle F., Bitsche R., Yde A., Henriksen LC., Natarajan A., et al. Description of the DTU 10 MW Reference Wind Turbine Department of Wind Energy I-Report. 2013. Available at: <https://dtu-10mw-rwt.vindenergi.dtu.dk>
13. Hau E. Wind Turbines - Fundamentals, Technologies, Application, Economics. Third. 1981. Available at: DOI:10.4324/9780203103289-9
14. Hayman B., Wedel-Heinen J., Brøndsted P. Materials Challenges in Present and Future Wind Energy. *MRS Bulletin*. 2008; 33(4): 343–353. Available at: DOI:10.1557/mrs2008.70
15. Hansen MOL. Aerodynamics of Wind Turbines. Third. London: Routledge; 2008.
16. Browning K. Commercial Wind Turbine Cost (\$1 Million Cost Breakdown). *Climatebiz*. 2022. Available at: <https://climatebiz.com/commercial-wind-turbine-cost/>
17. Molly J-P. Wind Energy : Theory - Application - Measurement. 2nd edn. 1990.
18. Schubel PJ., Crossley RJ. Wind Turbine Blade Design. *Energies*. 2012; 5(9): 3425–3449. Available at: DOI:10.3390/en5093425
19. Hansen MOL., Sørensen JN., Voutsinas S., Sørensen N., Madsen HA. State of the Art in Wind Turbine Aerodynamics and Aeroelasticity. *Progress in Aerospace Sciences*. 2006; 42(4): 285–330. Available at: DOI:10.1016/j.paerosci.2006.10.002

20. J. G. McGowan, R. W. Hyers, K. L. Sullivan, J. F. Manwell, S. V. Nair BM &B. CS. A Review of Materials Degradation in Utility Scale Wind Turbines. *Materials Science and Engineering for Energy Systems*. 2007; 2(1): 41–64.
21. Dominy RG., Lunt P., Bickerdyke A., Dominy J. Self-Starting Capability of a Darrieus Turbine. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*. 2007; 221(1): 111–120. Available at: DOI:10.1243/09576509JPE340
22. Burton T., Jenkins N., Sharpe D., Bossanyi E. *Wind Energy Handbook*. John Wiley & Sons, Ltd. 2011. 1–542 p.
23. Gasch R., Twele J. *Wind Power Plants: Fundamentals, Design, Construction and Operation*. Second. Springer; 2012. 548 p. Available at: http://books.google.com/books?id=c-QB9PiC_GMC&pgis=1
24. Gorban' AN., Gorlov AM., Silantyev VM. Limits of the Turbine Efficiency for Free Fluid Flow. *Journal of Energy Resources Technology, Transactions of the ASME*. 2001; 123(4): 311–317. Available at: DOI:10.1115/1.1414137
25. Yurdusev MA., Ata R., Çetin NS. Assessment of Optimum Tip Speed Ratio in Wind Turbines Using Artificial Neural Networks. *Energy*. 2006; 31(12): 2153–2161. Available at: DOI:10.1016/j.energy.2005.09.007
26. Hull DG. *Fundamentals of Airplane Flight Mechanics*. Springer. Springer; 2007. Available at: https://scholar.google.co.uk/scholar?q=fundamental+of+airplane+flight+mechanics+david+hull&btnG=&hl=en&as_sdt=0,5#0
27. Castelli MR., Benini E. Comparison between Lift and Drag-Driven VAWT Concepts on Low-Wind Site AEO. *International Journal of Environmental and Ecological Engineering*. 2011; 5(11): 669–674. Available at: <https://doi.org/10.5281/zenodo.1083101>
28. Grasso F. Development of Thick Airfoils for Wind Turbines. *Journal of Aircraft*. 2013; 50(3): 975–981. Available at: DOI:10.2514/1.C032182
29. Giebel G., Al. E. The State of the Art in Short-Term Prediction of Wind Power. *Proc. of SeaTechWeek, Brest, France*. 2004. pp. 1–13.

30. Pinson P., Chevallier C., Kariniotakis G. Optimizing Benefits from Wind Power Participation in Electricity Markets using Advanced Tools for Wind Power Forecasting and Uncertainty Assessment. EWEC Conference. 2004. pp. 1–8.
31. Wilson RE., P.B.S. L. Applied Aerodynamics of Wind Power Machines. 1974.
32. Barlas TK., van Kuik GAM. Review of State of the Art in Smart Rotor Control Research for Wind Turbines. Progress in Aerospace Sciences. Elsevier; 2010; 46(1): 1–27. Available at: DOI:10.1016/j.paerosci.2009.08.002
33. Rahimi M. Control and Performance Assessment of Variable Rotor Resistance Based Wind Turbines Regarding the Aerodynamic Power Fluctuations. Scientia Iranica. 2017; 25: 1593–1607. Available at: DOI:10.24200/sci.2017.4363
34. James F. Manwell, Jon G. McGowan AL. Wind Energy Explained Theory, Design and Application. Second. WILEY; 2010. 704 p. Available at: https://books.google.co.uk/books?id=roaTx_Of0vAC&printsec=frontcover#v=onepage&q&f=true
35. Christian Haag K. System and Method for Pitching a Rotor Blade in A Wind Turbine. United States Patent; 2013.
36. Polinder H., Van Der Pijl FFA., De Vilder GJ., Tavner PJ. Comparison of Direct-Drive and Geared Generator Concepts for Wind Turbines. IEEE Transactions on Energy Conversion. 2006; 21(3): 725–733. Available at: DOI:10.1109/TEC.2006.875476
37. Duquette MM., Visser KD. Numerical Implications of Solidity and Blade Number on Rotor Performance of Horizontal-Axis Wind Turbines. Journal of Solar Energy Engineering, Transactions of the ASME. 2003; 125(4): 425–432. Available at: DOI:10.1115/1.1629751
38. Oerlemans S., Sijtsma P., Méndez López B. Location and Quantification of Noise Sources on a Wind Turbine. Journal of Sound and Vibration. 2007; 299: 869–883. Available at: DOI:10.1016/j.jsv.2006.07.032
39. Chattot JJ. Optimization of Wind Turbines Using Helicoidal Vortex Model. Journal of Solar Energy Engineering, Transactions of the ASME. 2003; 125(4):

- 418–424. Available at: DOI:10.1115/1.1621675
40. Thresher RW., Dodge DM. Trends in the Evolution of Wind Turbine Generator Configurations and Systems. *Wind Energy*. 1998; 1: 70–85. Available at: DOI:10.1002/(sici)1099-1824(199804)1:1+<70::aid-we2>3.3.co;2-0
 41. Gipe P. The Wind Industry's Experience with Aesthetic Criticism. *IEEE Xplore*. 1990;
 42. Chamberlain DE., Rehfisch MR., Fox AD., Desholm M., Anthony SJ. The Effect of Avoidance Rates on Bird Mortality Predictions Made by Wind Turbine Collision Risk Models. *British Ornithologists' Union, Ibis*,. 2006; 148: 198–202. Available at: DOI:10.1111/j.1474-919X.2006.00507.x
 43. Maalawi KY., Badr MA. A Practical Approach for Selecting Optimum Wind Rotors. *Renewable Energy*. 2003; 28(5): 803–822. Available at: DOI:10.1016/S0960-1481(02)00028-9
 44. Drela M. An Analysis and Design System for Low Reynolds Number Airfoils. *Low Reynolds Number Aerodynamics*. Springer-Verlag; 1989. p. 12. Available at: http://link.springer.com/chapter/10.1007/978-3-642-84010-4_1
 45. Timmer WA., Van Rooij RPJOM. Summary of the Delft University Wind Turbine Dedicated Airfoils. *Journal of Solar Energy Engineering, Transactions of the ASME*. 2003; 125(4): 488–496. Available at: DOI:10.1115/1.1626129
 46. Van Rooij RPJOM., Timmer WA. Roughness Sensitivity Considerations for Thick Rotor Blade Airfoils. *Journal of Solar Energy Engineering, Transactions of the ASME*. 2003; 125(4): 468–478. Available at: DOI:10.1115/1.1624614
 47. Fuglsang P., Bak C. Development of the Risø Wind Turbine Airfoils. *Wind Energy*. 2004; 7(2): 145–162. Available at: DOI:10.1002/we.117
 48. Gupta S., Leishman JG. Dynamic Stall Modelling of the S809 Aerofoil and Comparison with Experiments. *Wind Energy*. 2006; 9(6): 521–547. Available at: DOI:10.1002/we.200
 49. Stol KA., Wenxin Z., Wright AD. Individual Blade Pitch Control for the Controls Advanced Research Turbine (CART). *Journal of Solar Energy Engineering, Transactions of the ASME*. 2006; 128(4): 498–505. Available at:

DOI:10.1115/1.2349542

50. Bossanyi EA. Individual Blade Pitch Control for Load Reduction. *Wind Energy*. 2003; 6(2): 119–128. Available at: DOI:10.1002/we.76
51. Kong C., Bang J., Sugiyama Y. Structural Investigation of Composite Wind Turbine Blade Considering Various Load Cases and Fatigue Life. *Energy*. 2005; 30: 2101–2114. Available at: DOI:10.1016/j.energy.2004.08.016
52. Corke T., Nelson R. Wind Turbine Control Methods. *Wind Energy Design*. 2020; : 135–160. Available at: DOI:10.1201/b22301-6
53. Schepers JG., Schreck SJ. Aerodynamic Measurements on Wind Turbines. Wiley : WIREs Energy and Environment. 2019; 8(1): 1–25. Available at: DOI:10.1002/wene.320
54. H. Aagaard Madsen SMP. Wind Turbine Test Tellus T-1995, 95 kW. Risoe National Lab., Roskilde (Denmark). Test Station for Wind Turbines. 1990; RISO-M-276: 68.
55. Sørensen NN. Transition Prediction on the NORDTANK 500 / 41 Turbine Rotor. Forskningscenter Risoe. 2002;
56. Johansen J., Sørensen NN. Aerofoil Characteristics from 3D CFD Rotor Computations. *Wind Energy*. 2004; 7(4): 283–294. Available at: DOI:10.1002/we.127
57. Rahimi H., Hartvelt M., Peinke J., Schepers JG. Investigation of the Current Yaw Engineering Models for Simulation of Wind Turbines in BEM and Comparison with CFD and Experiment. *Journal of Physics: Conference Series*. 2016; 753(2). Available at: DOI:10.1088/1742-6596/753/2/022016
58. Rahimi H., Schepers JG., Shen WZ., García NR., Schneider MS., Micallef D., et al. Evaluation of Different Methods for Determining the Angle of Attack on Wind Turbine Blades with CFD Results Under Axial Inflow Conditions. *Renewable Energy*. Elsevier Ltd; 2018; 125: 866–876. Available at: DOI:10.1016/j.renene.2018.03.018
59. Rahimi H., Dose B., Stoevesandt B., Peinke J. Investigation of the Validity of BEM for Simulation of Wind Turbines in Complex Load Cases and Comparison

- with Experiment and CFD. *Journal of Physics: Conference Series*. 2016; 749(1). Available at: DOI:10.1088/1742-6596/749/1/012015
60. van Rooij RPJOM. Modification of the Boundary Layer Calculation in RFOIL for Improved Airfoil Stall Prediction. 1996.
 61. Gault DE. A Correlation of Low-speed, Airfoil-Section Stalling Characteristics with Reynolds Number and Airfoil Geometry. NACA Report. 1957.
 62. Méndez B., Munduate X., Miguel US. Airfoil Family Design for Large Offshore Wind Turbine Blades. *Journal of Physics: Conference Series*. 2014; 524(1). Available at: DOI:10.1088/1742-6596/524/1/012022
 63. Parasaram RKN., Charyulu TN. Airfoil Profile Design By Reverse Engineering Bezier Curve. *International Journal of Mechanical Engineering and Robotics Research*. 2012; 1(3): 410–420.
 64. Gómez-Iradi S., Steijl R., Barakos GN. Development and Validation of a CFD Technique for the aerodynamic Analysis of HAWT. *Journal of Solar Energy Engineering, Transactions of the ASME*. 2009; 131(3): 1–13. Available at: DOI:10.1115/1.3139144
 65. Zhu WJ., Shen WZ., Sørensen JN. Integrated Airfoil and Blade Design Method for Large Wind Turbines. *International Conference on Aerodynamics of Offshore Wind Energy Systems and wakes (ICOWES2013)*. 2013. Available at: DOI:10.1016/j.renene.2014.02.057
 66. Muñoz A., Méndez B., Munduate X. Thick Airfoil Designs for the Root of the 10MW INNWIND.EU Wind Turbine. *Journal of Physics: Conference Series*. 2016; 753(2). Available at: DOI:10.1088/1742-6596/753/2/022046
 67. Brand AJ., Peinke J., Mann J. Turbulence and Wind Turbines. *European Turbulence Conference, Warsaw, Poland*. 2011. Available at: DOI:10.1088/1742-6596/318/7/072005
 68. Mücke T., Kleinhans D., Peinke J. Atmospheric Turbulence and Its Influence on the Alternating Loads on Wind Turbines. *Wind Energy*. 2011; 14: 301–316. Available at: DOI:10.1002/we
 69. Höyland J. Challenges for Large Wind Turbine Blades. 2010.

70. Giahi MH., Jafarian Dehkordi A. Investigating the Influence of Dimensional Scaling on Aerodynamic Characteristics of Wind Turbine Using CFD Simulation. *Renewable Energy*. Elsevier Ltd; 2016; 97: 162–168. Available at: DOI:10.1016/j.renene.2016.05.059
71. Griffith DT., Ashwill TD. The Sandia 100-meter All-glass Baseline Wind Turbine Blade : SNL100-00. Sandia National Laboratories. 2011.
72. Shimizu Y., Kamada Y. Studies on a Horizontal Axis Wind Turbine with Passive Pitch-Flap Mechanism (Performance and Flow Analysis Around Wind Turbine). *Journal of Fluids Engineering, Transactions of the ASME*. 2001; 123(3): 516–522. Available at: DOI:10.1115/1.1375817
73. Gaunaa M., Johansen J. Determination of the Maximum Aerodynamic Efficiency of Wind Turbine Rotors with Winglets. *Journal of Physics: Conference Series*. 2007; 75(1). Available at: DOI:10.1088/1742-6596/75/1/012006
74. Whitcomb RT. A Design Approach and Selected Wind Tunnel Results at High Subsonic Speeds for Wing-Tip Mounted Winglets. Nasa Tn D-8260. 1976; : 1–33. Available at: <https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19760019075.pdf>
75. Eppler R. Induced Drag and Winglets. *Aerospace Science and Technology*. 1997; 1(1): 3–15. Available at: DOI:10.1016/S1270-9638(97)90019-5
76. Maughmer MD. Design of Winglets for High-Performance Sailplanes. *Journal of Aircraft*. 2003; 40(6): 1099–1106. Available at: DOI:10.2514/1.10817
77. Farhan A., Hassanpour A., Burns A., Motlagh YG. Numerical Study of Effect of Winglet Planform and Airfoil on a Horizontal Axis Wind Turbine Performance. *Renewable Energy*. Elsevier Ltd; 2019; 131: 1255–1273. Available at: DOI:10.1016/j.renene.2018.08.017
78. M.M. Hand, D.A. Simms, L.J. Fingersh, D.W. Jager, J.R. Cotrell, S. Schreck SML. Unsteady Aerodynamics Experiment Phase VI: Wind Tunnel Test Configurations and Available Data Campaigns. National Renewable Energy Laboratory. 2001.

79. Maughmer MD. The Design of Winglets for High-Performance Sailplanes. AIAA -2406. 2001;
80. Zahle F., Sørensen NN., McWilliam MK. Computational Fluid Dynamics-Based Surrogate Optimization of a Wind Turbine Blade Tip Extension for Maximising Energy Production. *Journal of Physics: Conference Series*. 2018; 1037. Available at: DOI:10.1088/1742-6596/1037/4/042013
81. Bortolotti P., Tarres HC., Dykes KL., Merz K., Sethuraman L., Verelst D., et al. Systems Engineering in Wind Energy - WP2.1 Reference Wind Turbines. National Renewable Energy Laboratory. 2019. Available at: <https://www.osti.gov/biblio/1529216-iea-wind-tcp-task-systems-engineering-wind-energy-wp2-reference-wind-turbines%0Ahttp://www.osti.gov/servlets/purl/1529216/>
82. Al-Abadi A., Kim Y., Ertunc., Epple P., Delgado A. Interaction between Free-Stream Turbulence and Tip-Vortices of Wind Turbine Blades with and without Winglets. *Journal of Physics: Conference Series*. 2018. Available at: DOI:10.1088/1742-6596/1037/7/072025
83. Kurt S. Hansen, Rebecca J. Barthelmie, Leo E. Jensen AS. The Impact of Turbulence Intensity and Atmospheric Stability on Power Deficits Due to Wind Turbine Wakes at Horns Rev Wind Farm. *Wind Energy*. 2012; 15: 183–196. Available at: DOI:10.1002/we
84. Jonkman J., Butterfield S., Musial W., Scott G. Definition of a 5-MW reference wind turbine for offshore system development. National Renewable Energy Laboratory. 2009. Available at: http://tethys-development.pnnl.gov/sites/default/files/publications/Jonkman_et_al_2009.pdf
85. Fabio Rizzo, Piero D'Asdia FS. Tourist Living on Off-Shore Wind Turbine: Floating Anchorage Design and Wind/Wave – Structure Interaction Study. *American Journal of Civil Engineering and Architecture*. 2015; 3(3): 101–108. Available at: DOI:10.12691/ajcea-3-3-6
86. Smith K. WindPACT Turbine Design Scaling Studies Technical Area 2: Turbine, Rotor, and Blade Logistics. National Renewable Energy Laboratory, NREL/SR-500 2001. Available at: <http://nrel.gov/docs/fy01osti/29439.pdf>

87. Sten Frandsen, Helge Gravesen, Lars Jørgensen CE. Recommendation for Technical Approval of Offshore Wind Turbines. 2001.
88. Bulder H., Hagg F., van Bussel G., Zaaier MB. Dutch Offshore Wind Energy Converter. 2000.
89. Sieros G., Chaviaropoulos P., Sørensen JD., Bulder BH., Jamieson P. Upscaling Wind Turbines: Theoretical and Practical Aspects and Their Impact on the Cost of Energy. *Wind Energy*. 2012; 15(1): 3–17. Available at: DOI:10.1002/we.527
90. Bjorck A. Coordinates and Calculations for the FFA-W1-xxx, FFA-W2-xxx and FFA-W3-xxx Series of Airfoils for Horizontal Axis Wind Turbines. The Aeronautical Research Institute of Sweden. 1990.
91. Fuglsang P., Thomsen K. Site-Specific Design Optimization of 1.5–2.0 MW Wind Turbines. *Journal of Solar Energy Engineering, Transactions of the ASME*. 2001; 123(4): 296–303. Available at: DOI:10.1115/1.1404433
92. Mateus MM., do Vale M., Rodrigues A., Bordado JC., Galhano dos Santos R. Is biomass liquefaction an option for the viability of poplar short rotation coppices? A preliminary experimental approach. *Energy*. 2017; 124: 40–45. Available at: DOI:10.1016/j.energy.2017.02.059
93. Gnanasekaran S., Saravanan N., Ilangkumaran M. Influence of injection timing on performance, emission and combustion characteristics of a DI diesel engine running on fish oil biodiesel. *Energy*. 2016; 116: 1218–1229. Available at: DOI:10.1016/j.energy.2016.10.039
94. Zhong W., An H., Shen L., Dai T., Fang W., Gao X., et al. Global Pattern of the International Fossil Fuel Trade: The Evolution of Communities. *Energy*. Elsevier Ltd; 2017; 123: 260–270. Available at: DOI:10.1016/j.energy.2017.02.033
95. Mertens S. The Energy Yield of Roof Mounted Wind Turbines. *Wind Engineering*. 2003; 27(6): 507–518. Available at: DOI:10.1260/030952403773617472
96. Van Kuik GAM., Peinke J., Nijssen R., Lekou D., Mann J., Sørensen JN., et al.

- Long-term Research Challenges in Wind Energy- A Research Agenda by the European Academy of Wind Energy. *Wind Energy Science*. 2016; 1(1): 1–39. Available at: DOI:10.5194/wes-1-1-2016
97. Ma P., Lien FS., Yee E. Coarse-Resolution Numerical Prediction of Small Wind Turbine Noise with Validation Against Field Measurements. *Renewable Energy*. Elsevier Ltd; 2017; 102: 502–515. Available at: DOI:10.1016/j.renene.2016.10.070
 98. Srinivas Guntur, Niels N. Sørensen, Scott Schreck LB. Modelling Dynamic Stall on Wind Turbine Blades Under Rotationally Augmented Flow Fields Srinivas. *Wind Energy*. 2010; : 1–15. Available at: DOI:10.1002/we
 99. Lynch CE. *Advanced CFD Methods for Wind Turbine Analysis*. 2011.
 100. Gharali K., Johnson DA. Numerical Modelling of an S809 Airfoil Under Dynamic Stall, Erosion and High Reduced Frequencies. *Applied Energy*. Elsevier Ltd; 2012; 93: 45–52. Available at: DOI:10.1016/j.apenergy.2011.04.037
 101. Versteeg HK., Malalasekera W. *Introduction to Computational Fluid Dynamics. Introductory Fluid Mechanics*. 2012. 343–416 p. Available at: DOI:10.1017/cbo9780511761348.011
 102. Sørensen JN. Aerodynamic Aspects of Wind Energy Conversion. *Annual Review of Fluid Mechanics*. 2011; 43: 427–448. Available at: DOI:10.1146/annurev-fluid-122109-160801
 103. Yu H., Thé J. Validation and Optimization of SST $k-\omega$ Turbulence Model for Pollutant Dispersion within a Building Array. *Atmospheric Environment*. 2016; 145: 225–238. Available at: DOI:10.1016/j.atmosenv.2016.09.043
 104. Revell W., Marianna H., Alistair B. DESider – A European Effort on Hybrid RANS-LES Modelling: Results of the European-Union Funded Project. 2007. Available at: DOI:10.1007/978-3-540-92773-0
 105. Blocken B., Stathopoulos T., van Beeck JPAJ. Pedestrian-Level Wind Conditions Around Buildings: Review of Wind-Tunnel and CFD Techniques and Their Accuracy for Wind Comfort Assessment. *Building and Environment*.

- Elsevier Ltd; 2016; 100: 50–81. Available at:
DOI:10.1016/j.buildenv.2016.02.004
106. Zhiyin Y. Large-Eddy Simulation : Past , Present and the Future. Chinese Journal of Aeronautics. 2015; 28(1): 11–24.
 107. Spalart PR., Allmaras SR. A One-Equation Turbulence Model for Aerodynamic Flows. 30th Aerospace Sciences Meeting & Exhibit. 1992; : 1–22. Available at: DOI:10.2514/6.1992-439
 108. Jou WH., Strelets MK., Allmaras SR., Spalart PR. Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach. Advances in DNS/LES. 1997. pp. 2–12. Available at:
<https://www.researchgate.net/publication/236888805%0AComments>
 109. Aranake AC., Lakshminarayan VK., Duraisamy K. Computational Analysis of Shrouded Wind Turbine Configurations Using a 3-Dimensional RANS Solver. 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition 2013. 2013; : 1–23. Available at:
DOI:10.2514/6.2013-1211
 110. Dose B. CFD Simulations of 2.5 MW Wind Turbine using ANSYS CFX and OpenFOAM. 2013. Available at: DOI:10.13140/RG.2.2.13292.26245
 111. Launder BE., Spalding DB. Lectures in mathematical models of turbulence. London, England: Academic Press; 1972.
 112. Yakhot V., Orszag SA. Renormalization Group Analysis of Turbulence. Journal of Scientific Computing. 1986; 1: 3–51. Available at: DOI:10.1007/BF01061452
 113. Tsan-Hsing Shih, William W. Liou, Aamir Shabbir ZY and JZ. A New k- ϵ Eddy Viscosity Model for High: Reynolds Number Turbulent Flows. Computers Fluids. 1995; 24(3): 227–238. Available at: DOI:10.1007/978-3-319-27386-0_7
 114. V Y., S.A. O., S. T., T.B. G., Speziale CG. Development of Turbulence Models for Shear Flows by a Double Expansion Technique. Physics of Fluids. 1992; : 5–29.
 115. Menter FR. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA Journal. 1994; 32(8): 1598–1605. Available at:

DOI:10.2514/3.12149

116. Menter FR., Kuntz M., Langtry R. Ten Years of Industrial Experience with the SST Turbulence Model. *Turbulence, Heat and Mass Transfer*. 2003; 4: 1–8.
117. ANSYS CFX-Solver Theory Guide. 2011.
118. Layer B. Basics of Y Plus, Boundary Layer and Wall Function in Turbulent Flows. LearnCAX. 2022. pp. 1–7. Available at:
<https://www.learncax.com/knowledge-base/blog/by-category/cfd/basics-of-y-plus-boundary-layer-and-wall-function-in-turbulent-flows>
119. Frei W. Which Turbulence Model Should I Choose for my CFD Application? COMSOL Blog. 2017; : 1–8. Available at: <http://www.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/>
120. Gilling L., Sørensen NN., Davidson L. Detached Eddy Simulations of an Airfoil in Turbulent Inflow. 47th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition. 2009; : 1–13. Available at:
DOI:10.2514/6.2009-270
121. Schulz C., Klein L., Weihsing P., Lutz T. Investigations into the Interaction of a Wind Turbine with Atmospheric Turbulence in Complex Terrain. *Journal of Physics: Conference Series*. 2016; 753. Available at: DOI:10.1088/1742-6596/753/3/032016
122. Sieros G., Chaviaropoulos P., Sørensen JD., Bulder BH., Jamieson P. Upscaling wind turbines: theoretical and practical aspects and their impact on the cost of energy. *Wind Energy*. 2012; 15(1): 3–17. Available at:
DOI:10.1002/we.527
123. Manwell JF., McGowan JG., Rogers AL. *Wind Energy Explained: Theory, Design and Application*. Wind Energy Explained: Theory, Design and Application. 2010. 105–132 p. Available at: DOI:10.1002/9781119994367
124. Hasager C., Vejen F. Disdrometers in Erosion. INNOVATIONSFONDEN.DK. 2017; : 1–9. Available at: <https://windenergy.dtu.dk/english/research/research-projects/erosion>
125. Ju SH., Huang YC., Huang YY. Study of optimal large-scale offshore wind

turbines. Renewable Energy. Elsevier Ltd; 2020; 154: 161–174. Available at:
DOI:10.1016/j.renene.2020.02.106

126. Siemens SWT-6.0-154 - Manufacturers and turbines - Online access - The Wind Power. Available at:
https://www.thewindpower.net/turbine_en_807_siemens_swt-6.0-154.php
(Accessed: 11 September 2022)

7. Appendix

7.1 Boundary Condition Interaction and Placement Sensitivity for Baseline Geometry

Different turbulence models and time settings were studied initially at 25 m/s , and the results are shown in Table 7.1. The k-omega SST model shows a significant improvement under steady state conditions, when compared with those from DTU and when compared with other turbulence models tested. As can be seen, the error is lower than 1% with this turbulence models while the error obtained with the other models are all above ~20% up to ~85%.

Table 7.1: Different turbulence models and time setting cases at velocity
25 m/s

	CFD	DTU	CFD	DTU	Percentage of error		
Models	C_t	C_t	C_p	C_p	C_t	C_p	Time
Laminar	0.027	0.06	0.034	0.043	-54.71	-19.79	Steady
Spalart-Allmaras	0.017	0.06	0.003	0.043	-72.22	-93.52	Steady
k-epsilon (standard)	0.024	0.06	0.006	0.043	-59.66	-84.91	Steady
k-epsilon (realizable)	0.031	0.06	0.017	0.043	-48.55	-61.08	Steady
k-omega SST	0.059	0.06	0.043	0.043	-0.993	-0.087	Steady
transition k-kl-omega	0.032	0.06	0.019	0.043	-46.33	-56.45	Steady
DES	0.044	0.06	0.026	0.043	-26.67	-39.54	Transient

7.2 Pitch Sensitivity Study for Baseline Geometry

Three different pitch angles were studied for a velocity of 5 *m/s*. The required thrust and power were achieved at pitch angle of 0.966 *degree*, where a low percentage of error (less than ~10%) was obtained.

Table 7.2: Percentage of error for thrust and power at velocity 5 *m/s*

	Pitch angle sensitivity study (CFD)		DTU 10 MW RWT		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
1.966	0.405	0.642	0.356	0.809	13.703	-20.676
0.966	0.376	0.728			5.628	-10.192
0.716	0.385	0.718			8.058	-11.434

Two different pitch angles were studied at 12 *m/s*. A low error on thrust and power were achieved with a pitch angle of 3.50 *degree* (error less than ~5%).

Table 7.3: Percentage of error for thrust and power at velocity 12 *m/s*

	Pitch angle sensitivity study (CFD)		DTU 10 MW RWT		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
4.50	1.206	9.485	1.325	11.170	-8.986	-15.082
3.50	1.392	11.501			5.076	2.962

For 16 m/s , two different pitch angles were considered. A low percentage of error of 1.125% and 0.178% were obtained for the required thrust and power respectively at a pitch angle of 11.50 *degree*.

Table 7.4: Percentage of error for thrust and power at velocity 16 m/s

	Pitch angle sensitivity study (CFD)		DTU 10 MW RWT		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
12.50	0.702	8.629	0.854	10.876	-17.740	-20.662
11.50	0.863	10.895			1.125	0.178

7.3 Pitch Sensitivity Study for Length Optimization

Table 7.5 and Table 7.6 show the pitch angle sensitivity study for the length optimization. The first simulation was carried out for the reduced pitch angle itself.

Table 7.5: Percentage of error for thrust and power at velocity 12 m/s

	Pitch angle sensitivity study (CFD)		QBlade		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
2.96	2.528	19.357	2.500	20.200	1.124	-4.174

Table 7.6: Percentage of error for thrust and power at velocity 20 *m/s*

	Pitch angle sensitivity study (CFD)		QBlade		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
12.66	1.212	18.466	1.405	20.200	-13.76	-8.586
12.56	1.343	19.423			-4.400	-3.859

7.4 Pitch Sensitivity Study for Chord Length and Twist Angle Optimization

Table 7.7 to Table 7.9 show the pitch angle sensitivity study for a velocity of 5 *m/s*, 12 *m/s* and 16 *m/s*.

Table 7.7: Percentage of error for thrust and power at velocity 5 *m/s*

	Pitch angle sensitivity study (CFD)		QBlade		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
0.22	0.79	1.44	0.76	1.70	3.408	-15.423

Table 7.8: Percentage of error for thrust and power at velocity 12 *m/s*

	Pitch angle sensitivity study (CFD)		QBlade		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
5	2.415	20.455	2.480	21.400	-2.626	-4.415

Table 7.9: Percentage of error for thrust and power at velocity 16 *m/s*

	Pitch angle sensitivity study (CFD)		QBlade		Percentage of error	
Pitch angle (degree)	Thrust (MN)	Power (MW)	Thrust (MN)	Power (MW)	Thrust	Power
11.42	1.558	19.410	1.680	21.500	-7.281	-9.721
11.17	1.655	20.719			-1.487	-3.630