

Design of a shrouded wind turbine for low wind speeds

J.D. Human
13127160

Dissertation submitted in partial fulfilment of the requirements for the degree Master of
Engineering at the Potchefstroom Campus of the North-West University

Supervisor: Prof. C.P. Storm
Co-Supervisor: Dr. J.J. Bosman

November 2014

Abstract

The use of renewable energy is promoted worldwide to be less dependent on fossil fuels and nuclear energy. Therefore research in the field is driven to increase efficiency of renewable energy systems.

This study aimed to develop a wind turbine for low wind speeds in South Africa. Although there is a greater tendency to use solar panels because of the local weather conditions, there are some practical implications that have put the use of solar panels in certain areas to an end. The biggest problem is panel theft. Also, in some parts of the country the weather is more suitable to apply wind turbines.

Thus, this study focused on the design of a new concept to improve wind turbines to be appropriate for the low wind speeds in South Africa. The concept involves the implementation of a concentrator and diffuser to a wind turbine, to increase the power coefficient. Although the wind turbine was not tested for starting speeds, the implementation of the shroud should contribute to improved starting of the wind turbine at lower wind speeds.

The configuration were not manufactured, but simulated with the use of a program to obtain the power production of the wind turbine over a range of wind speeds. These values were compared to measured results of a open wind turbine developed for South Africa.

The most important matter at hand when dealing with a shrouded wind turbine is to determine if the overall diameter or the blade diameter of the turbine should be the point of reference. As the wind turbine is situated in a shroud that has a larger diameter than the turbine blades, some researchers believe that the overall diameter should be used to calculate the efficiency.

Theory was revised to determine the available energy in the shroud after initial calculations showed that the power coefficients should have been higher than the open wind turbine with the same total diameter. A new equation was derived to predict the available energy in a shroud.

The benefits of shrouded wind turbines are fully discussed in the dissertation content.

Keywords : Wind Turbine, Power Coefficient(C_p), Wind speed , Air speed in shroud

Opsomming

Die gebruik van hernubare energie word wêreldwyd gepromoveer ten einde die afhanklikheid aan fossielbrandstowwe en kernenergie te verminder. Dus word baie navorsing in hierdie gebied gedoen in 'n poging om die effektiwiteit van hernubare energiestelsels te verhoog.

Hierdie studie was daarop gerig om 'n windturbine te ontwikkel vir die lae windsnelheid in Suid-Afrika. Die weersomstandighede in Suid-Afrika lei egter tot 'n neiging om sonpanele vir energie opwekking te implementeer. Sekere, praktiese implikasies in spesifieke areas bemoeilik egter die uitsluitlike gebruik van sonpanele. Die grootste probleem is diefstal van die panele. Sekere streke in Suid-Afrika is ook meer geskik vir die gebruik van windturbines.

Die studie het gefokus om 'n nuwe konsep te ontwerp vir die lae wind snelhede in Suid-Afrika. Die konsep behels die implementering van 'n konsentreerder en diffusor aan 'n wind turbine om die krag koeffisiënt te verhoog. Alhoewel die wind turbine nie vir beginsnelhede getoets is nie, behoort die implementering van die huls ook by te dra tot die verlaging van die begin rotasie snelheid by laer wind snelhede.

Die konfigurasie is nie fisies gebou en opgerig nie, maar gesimuleer deur die toepassing van 'n program om waardes te verkry. Hierdie waardes is ook met vooraf gemete waardes van 'n oop wind turbine vergelyk, om te bepaal of die nuwe konfigurasie meer krag genereer as die oop tipe.

Die grootste kwessie betreffende die nuwe wind turbine is om te bepaal wat as verwysingspunt moet dien, naamlik die totale diameter of die turbine se lem diameter. Omrede die turbine in 'n huls is wat oor 'n groter diameter as die turbine lem beskik, meen sommige navorsers dat die totale diameter as verwysingspunt gebruik moet word ten einde die krag koeffisiënt te verkry.

Die teorie wat aanvanklik gebruik is om die beskikbare energie in die gehulde wind turbine te bepaal is hersien, nadat dit bevind is dat die energie te hoog voorspel is. 'n Nuwe vergelyking is afgelei om die beskikbare energie in 'n gehulde wind turbine te voorspel.

Hierdie ontwerp hou voordele in wat in detail in die studie bespreek word.

Sleutelwoorde: Windturbine, Kragkoeffisiënt(C_p), Windsnelheid, Lugsnelheid in huls

Acknowledgements

Thanks to:

- My Heavenly Father
- My dearest wife Christa for her love and support.
- My children Durandt and Annabell for making me smile and treasure life.
- Prof. C.P. Storm who helped realize a dream.
- Dr. J.J. Bosman for al his technical support.
- Albert Kriel for being an encouraging friend.

Contents

Nomenclature	viii
1 Introduction	1
1.1 Introduction	1
1.2 Problem Statement	2
1.3 Objective	2
1.4 Methodology	3
2 Literature Survey	5
2.1 Introduction	5
2.2 Aerodynamics	5
2.3 Wind turbine performance	6
2.3.1 Momentum theories	6
2.3.2 Induction factor	7
2.3.3 Tip and root losses	8
2.3.4 Solidity	8
2.3.5 Starting at low wind speeds	10
2.3.6 Reynolds number effects	11
2.3.7 Airfoil and blade design	12
2.4 Diffusers	13
2.5 Concentrators	18
2.6 Computational fluid dynamics (CFD)	19
2.7 Summary and proposed configuration	20
2.7.1 Summary	20
2.7.2 Proposed configuration	20
3 Theoretical background and CFD simulation setup	22
3.1 Introduction	22
3.2 Available power	22

3.3	Total power available in a shroud	24
3.4	Blade design	25
3.5	CFD simulation set-up	26
3.6	Summary	27
4	Validation	28
4.1	Introduction	28
4.2	Criteria for meaningful CFD results	28
4.3	Case 1: Diffuser	29
4.3.1	Diffuser parameters	29
4.3.2	Diffuser simulation set-up	29
4.3.3	Proses followed to reach cell independence for the diffuser simulations	31
4.3.4	Reflection on results and CFD solve information of the diffuser simulation . .	32
4.3.5	Summary of the diffuser simulations	33
4.4	Case 2: Open wind turbine	34
4.4.1	Wind turbine parameters	34
4.4.2	Wind turbine simulation set-up	35
4.4.3	Proses followed to reach cell independence for the wind turbine simulations .	36
4.4.4	Reflection on results and CFD solve information of the wind turbine simulations	39
4.4.5	Summary of the wind turbine simulations	40
4.5	Summary	40
5	Design	42
5.1	Introduction	42
5.2	Shroud design	42
5.2.1	Parameters for the shroud design	42
5.2.2	Shroud simulation set-up	43
5.2.3	CFD analyses of the shroud design	44
5.2.4	CFD solve information of the shroud design	45
5.2.5	Summary of the shroud design	45
5.3	Blade design	46
5.3.1	Introduction to the blade design	46
5.3.2	Parameters for the turbine blade design	46
5.3.3	Blade element theory design	48
5.3.4	Wind turbine simulation set-up	49
5.3.5	CFD analyses of the wind turbine	49
5.3.6	CFD solve information for the wind turbine design	51

5.3.7	Summary on blade design	51
6	Results and supportive theory	53
6.1	Introduction	53
6.2	Results	53
6.3	Available energy and mass flow	54
6.4	Calculated available power and new C_p values	57
6.5	Available power for the shrouded wind turbine and a wind turbine with the same turbine diameter	58
6.6	Reflections on results	58
6.7	Summary	60
7	Conclusions and Recommendations	62
7.1	Conclusions	62
7.2	Recommendations and Future studies	63
A	Validation diagrams and figures	69
B	Design diagrams and figures	73
C	Results and supportive theory	80

List of Figures

1.1	Wind turbine's incorporating a concentrator or diffuser	3
2.1	Two dimensional airfoil with labelled terminology	6
2.2	Power coefficient versus tip speed ratio for an ideal horizontal axis wind turbine . . .	7
2.3	Angular(a') and axial(a) induction factors for an ideal wind turbine	8
2.4	C_p values as a function of C_l/C_d ratio of a three-blades optimum turbine	9
2.5	Effect of solidity on $C_{p,Max}$ values	10
2.6	Blade number effects on C_p in a shrouded wind turbine	11
2.7	A 500W wind turbine power curve	12
2.8	Starting wind speed of a 500W wind turbine	13
2.9	Lift to Drag ratio of two types of airfoils with the top one lifted one unit	14
2.10	Annular and conical diffusers	15
2.11	DAWT with inlet shroud and brim	15
2.12	Power coefficient vs tip speed ratio of a wind turbine with brim	16
2.13	Velocity increase with different configurations of components and length ratio's . . .	17
2.14	Concentrator in a wind tunnel	18
2.15	Concentrator, Diffuser type of wind turbine	19
3.1	Actuator disk model for a wind turbine	23
3.2	Velocities for a cross-section blade element at radius r	25
3.3	Boundaries for the simulation domain in CFD	26
4.1	Schematic of a wind turbine equipped with a flanged diffuser shroud	29
4.2	Wind velocity distribution on the central axis of a circular-diffuser with different brim heights	30
4.3	Simulation domain for the diffuser without a wind turbine	30
4.4	Velocity(magnitude) for the diffuser without wind turbine	32
4.5	Wind velocity(magnitude) distribution on the central axis of the diffuser and Wall Y+ values	33

4.6	Air velocity(magnitude) plot on the central axis of the diffuser model in CFD and measured values	34
4.7	Domain for the simulation of the open wind turbine	36
4.8	Plane section through the center of the domain showing the velocity(magnitude) and volume mesh	37
4.9	Surface mesh of the wind turbine model, interfaces and inlet boundary	38
4.10	Power curve for simulated values and measured values for the open wind turbine . .	39
4.11	Plane section of one blade perpendicular with the radial direction	40
4.12	Wall Y^+ values on the surface of the blades	41
5.1	Diffuser dimensions	43
5.2	Available power as the inner concentrator radius increased with a decrease in flow area inside the diffuser	45
5.3	Velocity(magnitude) in the shrouded diffuser with brim and revolved airfoil concentrator	46
5.4	Velocity in shroud with airfoil concentrator moved towards the inlet	47
5.5	Cord of the designed blade	49
5.6	Twist of the designed blade	49
5.7	Variation of blade angle at a $3.5m/s$ free wind speed	50
5.8	Velocity plot of shrouded wind turbine @ $3.5m/s$	52
5.9	Blades in shroud	52
6.1	Results for the new wind turbine configurations and test results for the <i>AE 1.0kW</i> wind turbine	54
6.2	Control volume with static pressure and velocities	55
6.3	Streamlines for the diffuser, concentrator configuration	56
6.4	Pressure and velocity relations in an empty diffuser	57
6.5	C_p versus tip speed for the scaled wind turbine	59
7.1	Compact diffuser with wind turbine	64
A.1	Monitor Plot of diffuser with brim	69
A.2	Residuals for the validation of a diffuser with brim	70
A.3	Plane section through the center of the domain showing the volume mesh of the diffuser model	70
A.4	Plane section through the center showing the mesh at the diffuser wall	71
A.5	Momentum monitor plot for the validation of a three bladed open wind turbine . . .	71
A.6	Residuals for the validation of a three bladed open wind turbine	72

B.1	Wall Y^+ values for the inner shroud	73
B.2	Residuals of the shroud design	75
B.3	Monitor plot for the diffuser design	75
B.4	AE 1.0kW Wind speed/Power Coefficient @ maximum efficiency	76
B.5	Two Dimensional airfoil C_l and C_d plots with Re	76
B.6	Two Dimensional airfoil C_l and C_d plots with Re	77
B.7	Blade design in spreadsheet	77
B.8	Wall Y^+ values of blades	78
B.9	Monitor plot @ 3.5 m/s wind speed	78
B.10	Residuals plot @ 3.5 m/s wind speed	79
C.1	Wind speed/Power AE 1.0kW wind turbine	80
C.2	Tip speed ratio/wind speed AE 1.0kW wind turbine	81

List of Tables

4.1	Predicted rotational speeds of the <i>AE 1.0kW</i> wind turbine at certain wind speeds	35
5.1	Total available power with increase of the radius of the concentrator	44
5.2	Simulation results for the designed blade angle	50
5.3	Simulation results for an increase of θ_p of 5° on the blade @ a free wind speed of 3.5 m/s	51
6.1	New C_p values determined with Equation (6.3) as maximum available power	57
6.2	Total available power for a shrouded and open wind turbine	58
B.1	Simulation results for an increase of θ_p of 5° on the blade @ a wind speed of 2 m/s	73
B.2	Simulation results for an increase of θ_p of 5° on the blade @ a wind speed of 3.5 m/s	74
B.3	Simulation results for an increase of θ_p of 5° on the blade @ a wind speed of 5m/s	74
B.4	Simulation results for an increase of θ_p of 5° on the blade @ a wind speed of 7m/s	74
B.5	Simulation results for an increase of θ_p of 5° on the blade @ a wind speed of 9m/s	74

Nomenclature

a	Axial induction factor	R	Blade tip radius, m
a'	Rotational induction factor	r	Radial co-ordinate along blade, m
A_0	Inlet area streamtube, m ²	R_0	Inlet radius of streamtube, m
A_∞	Outlet area streamtube, m ²	R_∞	Outlet radius of streamtube, m
A_s	Area at the maximum velocity in shroud, m ²	Re	Reynolds number
c	Blade chord, m	S	Area at actuator disk, m ²
c	Power, W	U_0, v_1	Undisturbed axial velocity, m/s
C_d	Two-dimensional drag coefficient	U_1	Velocity in front of the wind turbine, m/s
C_l	Two-dimensional lift coefficient	U_2	Velocity at the back of the wind turbine, m/s
C_p	Power coefficient	U_∞, v_2	wind speed in the far-wake, m/s
$C_{p,Max}$	Maximum Power coefficient	U_s	Maximum velocity in shroud, m/s
D	Inlet diameter of diffuser, m	U_T	Total velocity at blade element, m/s
E	Work done by force on actuator disk, J	U_{Max}	Maximum Axial velocity in shroud, m/s
F	Force on actuator disk, N	W, P	Power, W
h	Brim height, m	<i>Greek symbols</i>	
L	Length of diffuser, m	α	Blade speed, rad
$\dot{m}$	Mass flow rate, kg · s ⁻¹	γ	Tip speed ratio
N	Number of blades	γ_r	Local tip speed ratio
p_0	Static pressure at control volume inlet, Pa	Ω	Blade speed, rad/s
p_1	Static in front of the wind turbine, Pa	ϕ	Blade inflow angle, rad
p_2	Static at the back of the wind turbine, Pa	ρ	density, kg · m ⁻³
p_s	Static at the maximum velocity in the shroud, Pa	θ_p	Blade twist angle, rad
P_w	Maximum available “kinetic” power, W	<i>Scripts</i>	
Q	Torque, Nm	0	Well upstream; undisturbed wind
		∞	Far-wake

Abbreviations

CAD Computer-Aided Design

CFD Computational Fluid Dynamics

DAWT Diffuser Augmented Wind Turbine

EES Engineering Equation Solver

HAWT Horizontal-axis wind turbines

RPM Revolutions per minute

Chapter 1

Introduction

1.1 Introduction

The worldwide increase in demand for energy and the obligation to protect the environment further necessitates the use of renewable energy. One such renewable energy resource that can be used is wind energy. The use of wind mills to produce energy from wind power dates back as far as 3000 years. From the late nineteenth century wind mills with generators (wind turbines) have been used to generate electricity (Burton et al. 2001, 1).

As the demand for energy increased, it became clear that it will be necessary to locate wind turbines at certain terrains and regions which previously have not been considered suitable. These terrains and regions may have gust, turbulence and low wind speeds or other physical constraints. Progressively more wind turbines tend to be installed at such complex terrains (Palma & F.A. Castro 2008). Also, recently more efficient designs have been introduced for low wind speeds as well as for urban use where turbulence, noise levels and appearance needed to be considered and addressed (Wright & Wood 2004). Some new designs propose that the turbine forms part of a building and/or structures. Other designs apply turbines in conjunction with solar panels or other types of renewable energy systems (Grant et al. 2008).

South Africa and most parts of Africa have a relative low average wind speed. The regions that do have a higher mean wind speed are small and usually confined to coastal areas and mountain escarpments (AFDB 2004). Wind near mountain escarpments and at a building environment generally has higher turbulence levels. This turbulence will have an effect on the performance of a wind turbine (Burton et al. 2001, 37 and 12). Thus, the design of a wind turbines should be adapted to reduce this influence. The urban environment greatly reduces the wind speed and thus also requires a efficient wind turbine to extract the maximum amount of energy from the slow moving air.

The need for electricity in most parts of the African regions accentuates the opportunity to introduce small wind turbines to produce energy in urban and rural areas where the average wind

speed is low. New designs should be considered to overcome obstacles to produce well needed energy. The power generated by a wind turbine is a fraction of the power available in the wind. This dimensionless parameter is referred to as the power coefficient (C_p). The C_p value is the ratio of the actual power produced to the power available in the wind (Wood 2011, 8). These new designs should have improved C_p values. Also, small wind turbines can be an alternative to other renewable energy products that may not be practical at certain locations.

1.2 Problem Statement

Most of the wind turbines that are on the market have been developed in countries that have higher mean wind speeds. These wind turbines do not work effectively in South African conditions. The imported wind turbines are designed to have high C_p values at higher wind speeds. These wind turbines will not generate much energy except for the period of time that the wind velocity is high. Also, a wind turbine that is optimised for high wind speeds usually have reduced efficiency at low wind speeds. These wind turbines will fail to start rotating at low wind speeds (Wood 2011, 101,119).

Locally designed wind turbines also face a similar problem. The design for low wind speeds also reflect on the performance at the occasion the wind speed is high. Small wind turbines do not have pitch adjustment and the blade will have non optimum angles of attack at wind speeds that was not the design wind speed (Wood 2011, 101). The available energy at low wind speed regions is a minimum, therefore the wind turbine should have high efficiencies at a wide range of wind speeds.

From this one can see the necessity for some new designs to enhance the C_p values of a wind turbine's rotor for low wind speeds regions. One way to increase the C_p value of the wind turbine is to use structures like concentrators and diffusers. Both of these configurations are impractical to use in high wind speed regions because of structural constraints (Wood 2011, 38). In low wind speed regions it could be feasible to use them to increase the C_p values of a wind turbine. It should be noted that these shrouded wind turbines will probably be practical for micro and small wind turbines only.

With a small, low wind speed wind turbine there is a even greater expectation to improve the C_p value, as the energy available is already minimal. To conclude it is evident that there is a definite need to improve the feasibility of small wind turbines in low wind speed conditions.

1.3 Objective

The main focus of this study was to design a new shrouded wind turbine configuration, which included a concentrator and/or diffuser as depicted in Figure 1.1. The incorporation of a diffuser

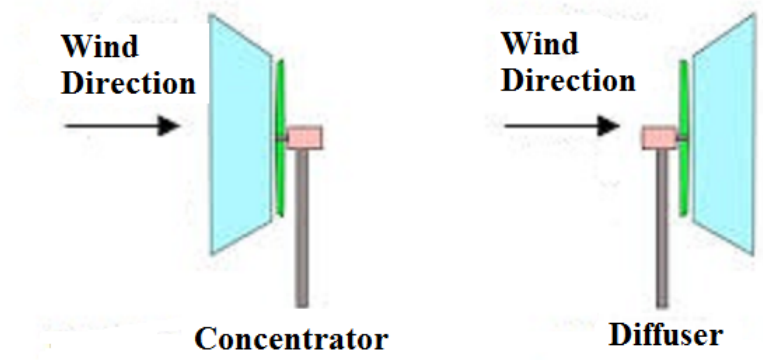


Figure 1.1: Wind turbine's incorporating a concentrator or diffuser

and concentrator to a wind turbine were implemented to increase the C_p values of a small wind turbine.

One of the objectives was also to compare this shrouded wind turbine with a wind turbine with the same total diameter. The total diameter of the shrouded wind turbine was chosen as $3.6m$. This diameter is the same as a wind turbine that was developed by Bosman et al. (2003) for wind speeds in South Africa. These two wind turbine's C_p values and power output were compared over a range of wind speeds.

The structural strength of this shrouded wind turbine configuration was not considered in this dissertation, as the aerodynamic design was the focus of this study. The power output at a number of wind speeds was calculated from torque values. These values were obtained from simulations in a Computational Fluid Dynamics(CFD) program. It was therefore not necessary to manufacture a prototype.

1.4 Methodology

A literature study is presented in Chapter 2. Blade design and the accompanied theories for horizontal axis wind turbines (HAWT) was investigated. The section also summarises the design and results obtained for a number of diffuser and concentrator types of wind turbines. The conclusion formed a basis for a new shrouded wind turbine configuration.

Chapter 3 presents a theoretical background on the available energy in the wind. The equations needed to design a blade with an elemental approach is shown in this chapter. The last section focus on the simulation set-up of the Computational Fluid Dynamics(CFD) simulations.

CFD was used to design the shrouded wind turbine. The process to validate the modelling of shrouds and wind turbines in CFD with experimental data is presented in Chapter 4.

The design chapter shows that the rotor was partially designed according to the strip or blade

element theory. CFD was used to complete the blade design and shroud design.

From the simulation results the generated power to wind speed curve (characteristic power performance curve) was obtained and was compared with a commercially available low wind speed turbine from South Africa. Chapter 6 also gives a reflection on results with supportive theory.

Chapter 2

Literature Survey

2.1 Introduction

This chapter opens with a short discussion of a few aerodynamics principles and the explanation of its basic terms. A further investigation to available literature on wind turbine design and performance (horizontal type) follows. Work on diffusers and concentrators is reflected upon. Computational fluid dynamics in general and guidelines for designing with the use of CDF is given. The chapter closes with a concise summary and a proposed shrouded wind turbine configuration.

2.2 Aerodynamics

Below follow definitions and explanations of a few aerodynamic principles and terms in order to understand later sections more clearly (see also Figure 2.1).

- **Drag on a two dimensional airfoil or body** is a force in the direction of the flow exerted on a body and can be divided into two parts, pressure drag and skin friction drag. The latter is drag due to shear stress. For example an infinite thin flat plate with the flow parallel over its surface will only experience friction drag. Pressure drag can be described by a plate oriented normal to the flow, the drag is due to the normal stress on the body. Total drag is therefore the combination of these two with a variation of the angle of attack (Shames 2003, 667).
- **A lift force on an turbine blade** can be determined by integrating the pressure force over the surface of the blade (Bertin & Cummings 2009, 215 and 216).
- From Figure 2.1 the **chord length** is the distance between the leading edge and the trailing edge. The angle of attack α is the angle between the relative air flow and the chord line. The **camber** is the asymmetry between the upper surface and lower surface of an airfoil.

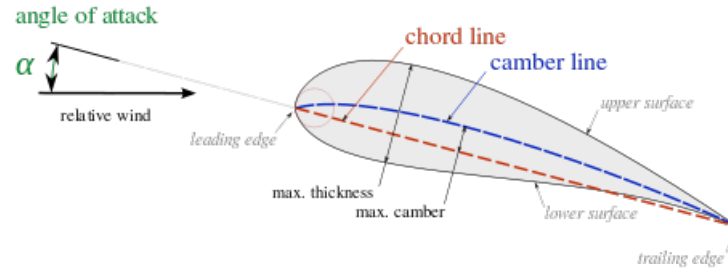


Figure 2.1: Two dimensional airfoil with labelled terminology

- **Separation** occurs when the fluid flow cannot follow the boundary layer at an adverse pressure gradient (Shames 2003, 666). In the case of an airfoil at high angles of attack α it is called a **stall condition** (Wood 2011, 60).

The wind turbine blade is a aerodynamic body, with the efficiency of the blade being effected by the aerodynamic performance.

2.3 Wind turbine performance

This section focuses on variables that may influence the design and performance of a wind turbine. For the most part the section concentrates on steady state performance which forms the basis for the development of a new wind turbine concept. It is also necessary to note that most small wind turbines do not have pitch adjustment (Wood 2011, 119) and therefore operate at variable rotational speed. Thus, the new concept for design will also follow the criteria set for variable rotational speed.

2.3.1 Momentum theories

One can not speak of wind turbine performance without mentioning a model, generally attributed to Betz in the 1930's (Manell et al. 2002, 84). It was based on a linear momentum theory developed to predict the performance of ship propellers. It predicts the maximum energy to be extracted from the free wind stream as a power coefficient (C_p) of 16/27 (0.5926) times that of the total available energy in the wind. This theory also allows for determining an induction factor that would predict the air velocity at the front of the blades as well as far downstream for maximum energy extraction.

The linear momentum theory assume that no rotation is imparted to the air flow. The turbine rotor rotates if a force (creates torque) is imparted on the blades by the air and thus an equal force on the air particles. The air particles behind the rotor will therefore have a tangential and an axial component (Burton et al. 2001, 47). Thus, there is an increase in the tangential kinetic energy that

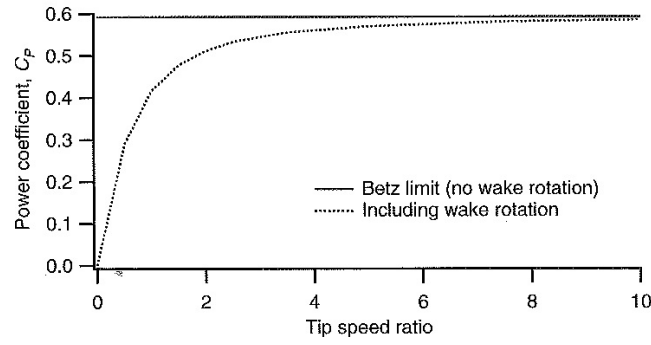


Figure 2.2: Power coefficient versus tip speed ratio for an ideal horizontal axis wind turbine (Manell et al. 2002, 94)

creates a wake with a drop in static pressure (Burton et al. 2001, 47).) in this wake. This can be seen as a loss in energy. The rotational kinetic energy that was imparted to the air will be higher if the torque imparted to the blade is higher. Thus a wind turbine with high rotational speed and low torque will experience fewer losses than a slow running one (Manell et al. 2002, 89). From this an equation can be derived to predict the power coefficient versus TSR (rotor tip tangent speed / free wind speed). The result of the equation is illustrated in Figure 2.2. Higher rotational speed (tip speed ratio) will give a increase in energy output by the wind turbine and low tip speed ratios should be avoided if possible. In sections 2.3.4 it is shown that a to high tip speed ratio can reduce the energy output of a wind turbine.

2.3.2 Induction factor

An axial induction factor, a , is the fractional decrease in air velocity between the free stream and the rotor plane. If a is equal to one third the turbine blade would extract the most energy from the wind (Manell et al. 2002, 86 to 87). From the angular momentum theory an induction factor, a' is the ratio between the angular velocity (wake) imparted to the stream divided by two times the angular velocity of the blades. As one should prefer a lower angular velocity imparted to the stream, it would suggest from the previous paragraph that for the most power output, the induction factor should be a minimum. The local speed ratio of a blade differs from the tip speed ratio. This will influence the induction factors near the hub. Figure 2.3 exemplify a turbine with a tip speed ratio (λ) of 7.5. It also shows that, closer to the hub the induction factors vary significantly. This will create low $C_{p,Max}$ values in this region near the hub. The influence of a and a' on the blade design can be seen in Figure 3.2.

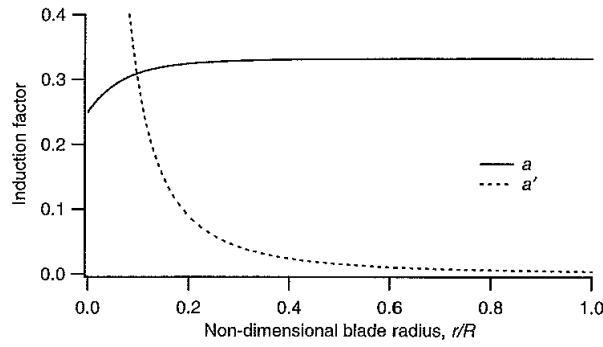


Figure 2.3: Angular(a') and axial(a) induction factors for an ideal wind turbine (Manell et al. 2002, 94)

2.3.3 Tip and root losses

Blade tip losses occur when air flow around the tip of the blade flow from the high static pressure side to the low pressure side. This effect reduces the lift force near the tip and hence the power production. The loss increases with fewer and wider blades (Manell et al. 2002, 118). The vortices that form on the tip of the blade are the same as the vortex on the tip of a wing of an aircraft wing (Wood 2011, 69) and have a great influence on the power production of wind turbines. With a shrouded type of wind turbine this loss can greatly be reduced.

Tip and hub vortices reduce the energy capture of the wind turbine (Manell et al. 2002, 142). Generally a loss factor is used to modify the produced torque of the wind turbine at the tip and root section and from this a new theoretical C_p curve can be drawn (Moriarty & Hansen 2005).

2.3.4 Solidity

Another principle matter to consider is solidity. It can be defined as the total blade area divided by the swept area. If the number of blades remained constant the blade chord should be altered to change the solidity. The solidity can also be changed by varying the number of blades (Manell et al. 2002, 174).

Figure 2.6 the observations evident when varying the number of blades with a **fixed blade angle** in order to alter the solidity. Low solidity produces a flat, broad C_p/λ curve. This means that the C_p will change very little over a variety of tip speeds. The $C_{p,Max}$ will be at a higher tip speed ratio but if the tip speed ratio is too high it will reduce the $C_{p,Max}$ value because of drag losses. Therefore a very efficient blade design with high lift to drag ratio is essential to obtain a high $C_{p,Max}$ value at a higher tip speed ratio (Figure 2.4). The solidity could also be increased through the introduction of more blades to get a higher $C_{p,Max}$ value and narrower C_p/λ curve (illustrated in Figure 2.6). In this case too high solidity will then reduce the $C_{p,Max}$ value as the blade will have a high angle of attack at lower tip speed ratio's with an increase of stall losses

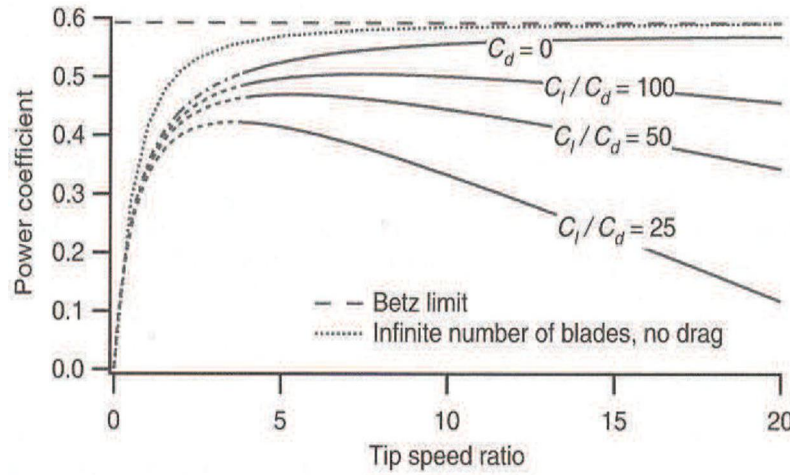


Figure 2.4: C_p values as a function of C_l/C_d ratio of a three-blades optimum turbine (Manell et al. 2002, 140)

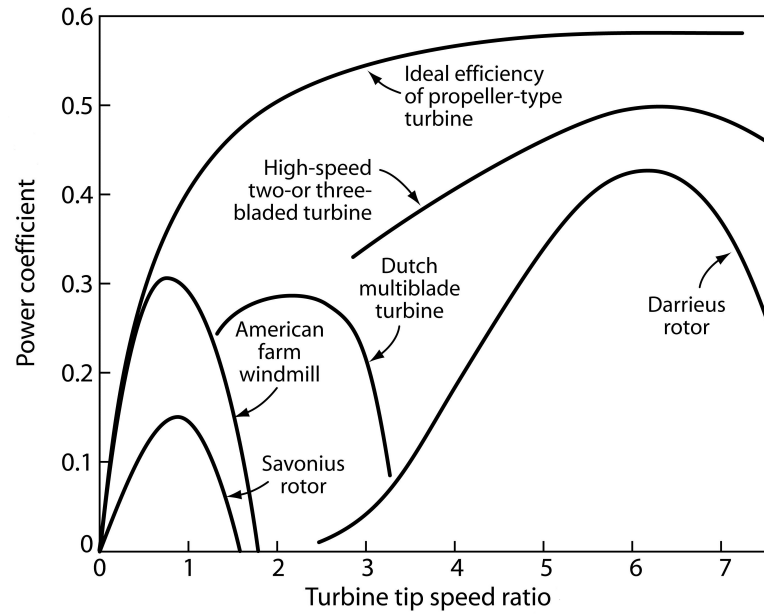
(Burton et al. 2001, 175).

At the root of the wind turbine the solidity is naturally very high that gives a lower $C_{p,Max}$ value in this region at a lower local speed ratio (γ_r). This lower local tip speed ratio is one of the reasons why the root section is responsible for starting the wind turbine as the torque produced is higher at a low γ and high solidity. Nearer to the tip of the blade the solidity becomes less and the local tip speed ratio increases with an increase in $C_{p,Max}$ values. This region is responsible for power production (Burton et al. 2001, 175). As high torque is necessary to pump water, a wind pump (American farm windmill) has high solidity, but this produces low $C_{p,Max}$ values as seen in Figure 2.5.

Wang & Chen (2008) concluded that higher blade numbers reduce starting wind speed as it creates higher starting torque. They used CFD to determine the number of blades to be used in a shrouded wind turbine. In the case of this specific blade design, Figure 2.6 shows that an optimum number of blades can improve C_p values, lower tip speed ratio and ultimately higher torque for starting as the tip speed ratio is lower. It should be noted that the $C_{p,Max}$ is high in the illustrated figure. This, can be attributed to the fact that the blade diameter, instead of the total diameter of the shroud, was taken as the reference diameter.

An optimum solution could be to apply a large number of blades with a short cord length, if structurally feasible (Burton et al. 2001, 175). However, it should be considered that the design of a hub for a large number of blades will possibly be troublesome.

Generators that requires higher torque when rotating or higher starting torque should have higher solidity wind turbines. There it would be a trade-off between having high $C_{p,Max}$ values, productivity (power production over a period of time) and starting at low wind speeds.

Figure 2.5: Effect of solidity on $C_{p,Max}$ values

2.3.5 Starting at low wind speeds

Very few small wind turbines have pitch adjustment. The blades with no pitch adjustment will therefore have very high angles of attack when stationary and even more if it was designed for a C_p values at high tip speed ratios. These high angles of attack at low Reynolds numbers make it difficult for the blade to produce a torque. To overcome the resistive torque of the generator and drive train micro turbines have five or more blades (Wood 2011, 101) to reduce the starting wind speeds.

Wright (2005) demonstrated from experimental measurements that the average starting wind speed is much higher than the cut-in speed. The cut in speed is the wind speed at which the wind turbine will deliver useful power (Manell et al. 2002, 7). In Figure 2.7 the power curve for a 500W wind turbine demonstrated a cut-in speed of $3.5m/s$ (Wood 2011, 102). The average starting speed ($4.8m/s$) is significantly higher for the same wind turbine as seen in Figure 2.8. If the starting wind speed is too high it will reduce the practicality or productivity (power production over a period of time) of the turbine (Wood 2011, 101). It is therefore necessary to keep starting speed in mind and not only high C_p values when designing a wind turbine.

Figure 3.2 confirms that if a high tip speed ratio is chosen for the design speed it results in a high ω that will increase the angle ϕ . This results in high angles of attack if starting and increase drag and low C_l values to produce a starting torque (Wood 2011, 64). It is important to note that low design tip speed ratio's can reduce the $C_{p,Max}$ values.

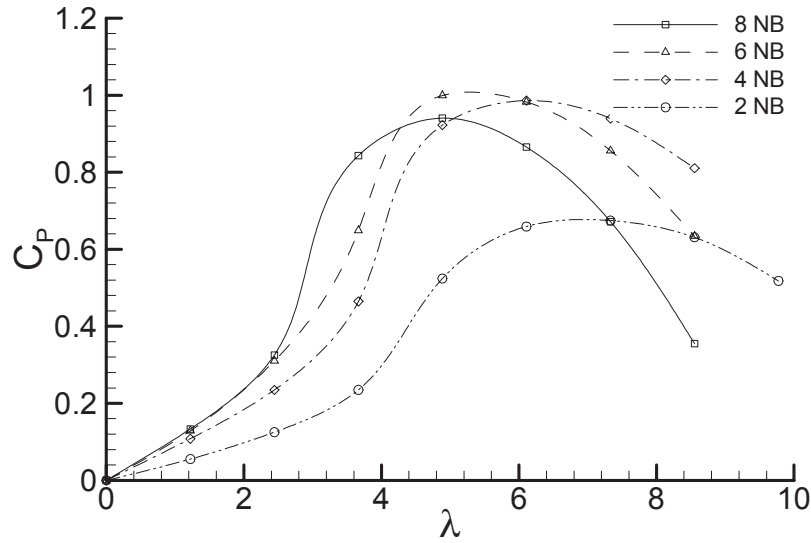


Figure 2.6: Blade number effects on C_p in a shrouded wind turbine (Wang & Chen 2008, 94)

2.3.6 Reynolds number effects

The Reynolds number for a two dimensional airfoil can be determined with chord length, density, dynamic viscosity and the total velocity the blade encounters (Wood 2011, 9). The lift and drag alter with angles of attack and Reynolds numbers (Wood 2011, 69), making Reynolds numbers important to consider.

The effects of low Reynolds numbers on small wind turbines are significant. With small wind turbines the drag is dominated by laminar separation which makes airfoil shape and design important (Giguère & Selig 1997). Although there is no fixed Reynolds number range, an airfoil performance roughly below 500,000 is primarily governed by a laminar separation bubble that forms on the surface which influences the performance of the wind turbine blade. The bubble gets smaller and the consequent drag decreases as the Reynolds number increases from 200,000 to 500,000. Reynolds numbers in the region of 70,000 and 200,000 could possibly achieve laminar flow without a bubble. Airfoil thickness has a great influence on bubble formations when Reynolds numbers are between 30,000 and 70,000 (Wood 2011, 71).

Thus, the airfoil selection forms a critical part of wind turbine design. Therefore the selection should be made for a specific range of Reynolds numbers considering that at these Reynolds numbers, variables like airfoil thickness influence airfoil performance (Giguère & Selig 1997).

The velocity of the air also has a significant influence on Reynolds numbers which leads to attempts with diffusers and concentrators to increase the effective air velocity at the blades.

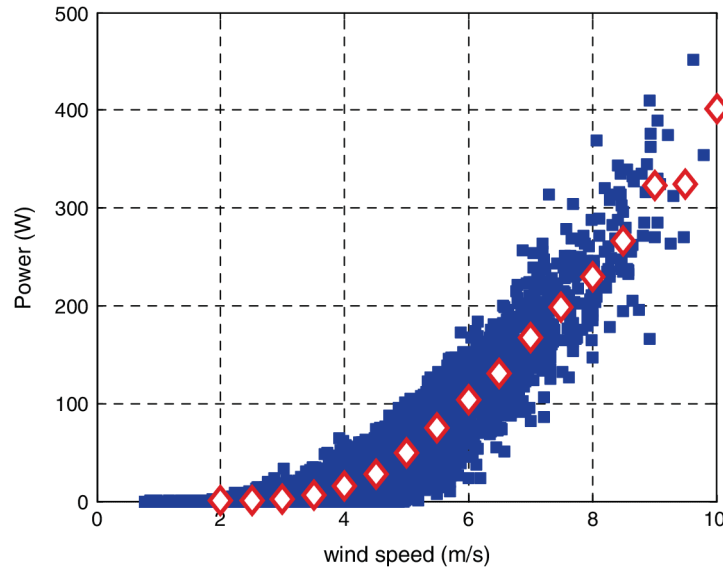


Figure 2.7: A 500W wind turbine power curve (Wright 2005)

2.3.7 Airfoil and blade design

There exists a simplified blade element theory in which it is assumed that the power (torque from lift) and thrust (from drag) that a wind turbine blades produces depend on the two dimensional lift and drag coefficients of the airfoil selected (Wood 2011, 57). The angle of attack (α) (Figure 2.1) together with the Reynolds number, influence this lift and drag coefficients and ultimately the power production from torque. Figure 2.9 shows how the lift/drag ratio is dependent of Reynolds numbers and angle of attack for a specific airfoil. A higher Reynolds number gives a better ratio, that also shows the benefit of increased air velocity.

For a specific blade design with a fixed design angle θ_p and α for the blade, the pressure drag (or form drag) on the blade will increase with a higher tip speed ratio as the inflow angle will become less favourable. This will greatly influence the performance of the blade as already mentioned in section 2.3.4 on solidity. If this blade design was made at a higher tip speed ratio the airfoil would have encountered stall losses at lower tip speed ratio's, as the relative velocity will influence the inflow angle.

It is difficult to determine the air speed at the front of the blades of a shrouded wind turbine as losses influence the incoming air speed substantially. Therefore Wang & Chen (2008) did some variation on the blade angle θ_p for a specific blade design. From this the $C_{p,Max}$ can be determined for a optimum tip speed ratio for the turbine in the diffuser. The $C_{p,Max}$ values for an optimum number of blades was also determined for the shrouded wind turbines, in CFD, as depicted in Figure 2.6.

The design of a wind turbine with no pitch adjustment implies that the design should be applied for variable speed rotational speed with a constant tip speed ratio. If the air speed at the front of the

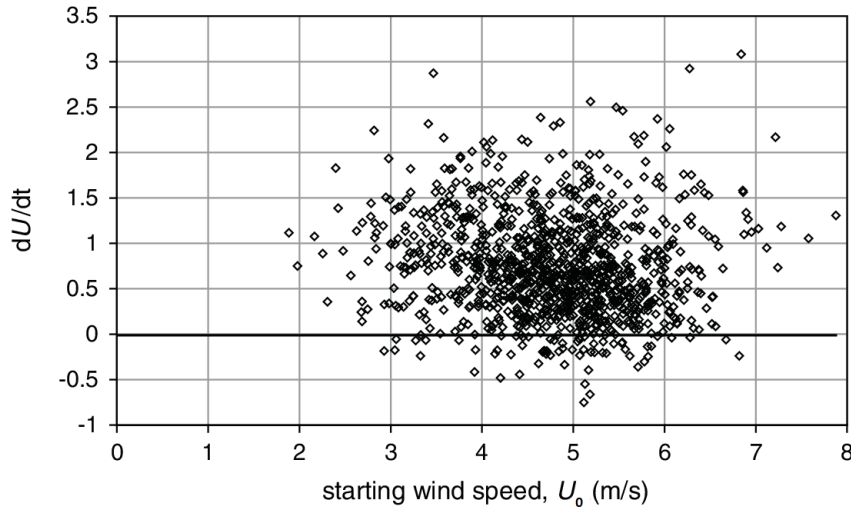


Figure 2.8: Starting wind speed of a 500W wind turbine (Wright 2005)

blades is known, then the number of blades, the two dimensional drag coefficient, two dimensional lift coefficient and tip speed ratio, the blade angles and chord lengths can be calculated. This can be used to calculate the torque and power production of the wind turbine. The blade element theory is generally used to design the blade of the wind turbine. For this theory the annular area of the turbine is divided in a number of smaller annular areas. The assumption of this theory is that the aerodynamic lift and drag forces exerted on a blade element in each of these annular areas is responsible for the change of momentum of the air and thus energy extraction (Burton et al. 2001, 59-77) .

2.4 Diffusers

A diffuser can be defined as a diverging passage which decelerates the flow of air with a rise in static pressure (Saravanamuttoo et al. 2001). There are mostly two types of axial diffusers in use, a diverging conical type and a conical annular type as illustrated in Figure 2.10.

From the previous paragraph it can be stated that a DAWT with the outlet of the diffuser at static atmospheric pressure would imply a lower static pressure at the inlet of the diffuser. As the static pressure of the atmosphere will be higher than the static pressure at the entrance of the diffuser, the air flow will increase and therefore the velocity at the entrance of the diffuser as well. Thus if a wind turbine is situated at the entrance of the diffuser it will encounter a higher air speed than the wind turbine without the diffuser.

Some early research had been done by Gilbert & Foreman (1983), Igra (1981) and Gilbert et al. (1978) on DAWT's. In their studies they concentrated the wind energy with a large open angle diffuser. Boundary layer control and separation were the main focuses as it could prevent

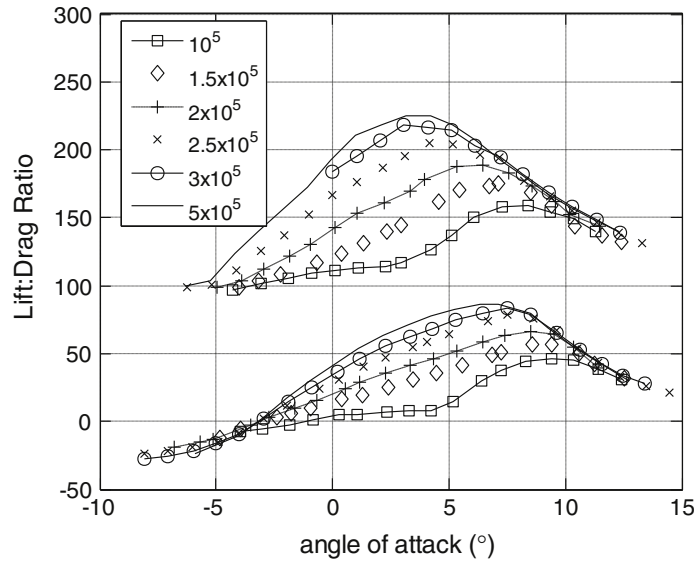


Figure 2.9: Lift to Drag ratio of two types of airfoils with the top one lifted one unit. Legend gives Reynolds numbers. (Wood 2011, 61)

pressure losses and increase the velocity inside the diffuser. The concept was never commercialized which indicated that it was not as profitable as researchers presumed. As Chen et al. (2011) later concluded on DAWT's, "The key problem in diffuser-augmented converters is to compensate at the outlet the pressure drop created by the turbine's energy extraction inside the duct". On the contrary they also revealed how DAWT's can be an attractive concept to apply. The outward deflection of the air flow on the outside of the diffuser creates a separation cavity at the end of the diffuser. This separation creates a low pressure region behind the diffuser. The lower outlet pressure will also produce a lower than atmospheric static pressure at the inlet of the diffuser. This lower pressure will increase the flow of air into the diffuser or can be used to create a greater pressure difference at the diffuser in a wind turbine. Both of these scenarios imply a higher energy output. It can thus be concluded that if this separation at the end of the shroud could be increased, it will increase the energy output of the wind turbine.

Matsushima et al. (2005) proposed a diffuser with a brim attached to its outlet. The brim increases the separation at the back of the diffuser and decreases the pressure near the outlet of the diffuser, at the back of the brim (this effect is the same as pressure drag, as earlier defined in section 2.2). Figure 2.11 shows a diffuser with brim and inlet shroud (in this design there was no inlet shroud). The illustration presents the large amount of separation at the rear of the shroud. This decreases the static pressure in this part of the diffuser. For this study a CFD simulation of a diffuser with an inlet diameter of $1m$, a total length of $2m$ to $4m$, a brim height of $100mm$ to $500mm$ and the angle of the diffuser between 0° and 12° was prepared by Matsushima et al. (2005). The simulation was done without a wind turbine. The air velocity ratio (between the free

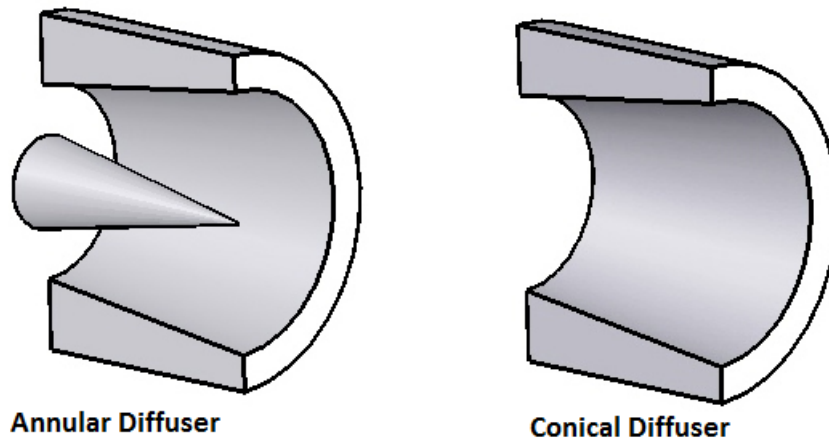


Figure 2.10: Annular and conical diffusers

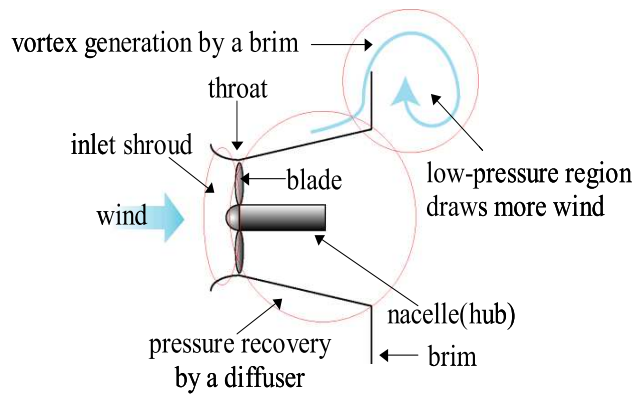


Figure 2.11: DAWT with inlet shroud and brim (Matsushima et al. 2005)

wind speed and inlet air speed) increased sharply from a diffuser angle of 0° to 4° and reached a maximum at 6° . It was found that the inlet velocity ratio did not increase at a brim height of more than a 100mm . An inlet velocity ratio of 1.7 was obtained with these dimensions.

The prototype built by Matsushima et al. (2005) had a diffuser length of 2m , diffuser angle of 4° and a brim height of a 100mm . A wind turbine with five blades was used inside the diffuser. The same type of wind turbine without a diffuser and brim was erected near this shrouded wind turbine to compare the energy output over a certain period of time. Some problems were experienced with the adjusting of the field device to the wind direction. Thus, the researchers fixed the conventional wind turbine and shrouded wind turbine in the direction where the frequency distribution of the wind was high. The total energy production for the entire day was measured and it was found that the shrouded wind turbine produced 1.65 times more energy than the conventional wind turbine.

Abe (2004) did a CFD investigation and found some important features for a diffuser with a

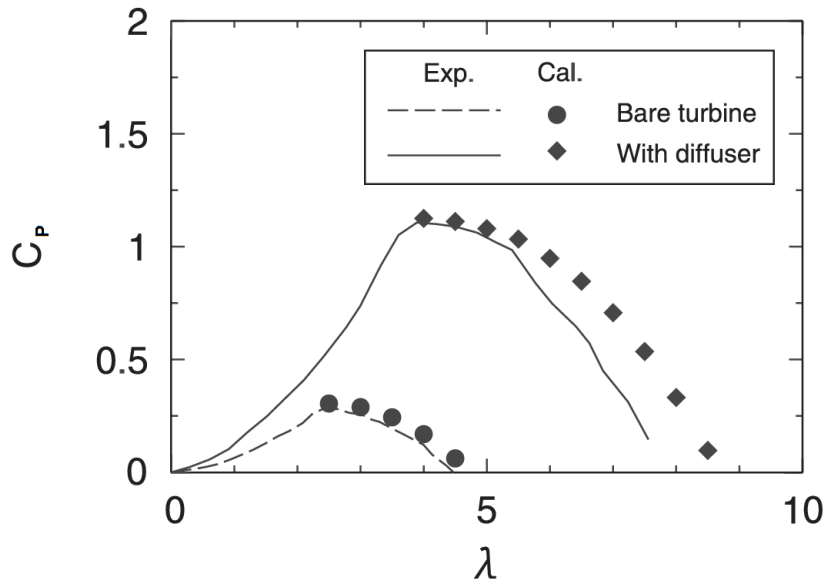


Figure 2.12: Power coefficient / tip speed ratio of a wind turbine with brim tested by Abe et al. (2005)

brim. The performance of the wind turbine depends strongly on the loading coefficient and the angle of the diffuser. This greatly affects the nature of the separation in the diffuser. From the investigation of Abe (2004) it was also clear that the loading coefficient should be much smaller than that of a normal HAWT's. If the loading coefficient is too high it will reduce velocity at the entrance and a higher pressure discontinuity over the wind turbine area. Therefore it will be possible to have a higher pressure drop over the area of the wind turbine with a lower air velocity or a lower pressure drop with a higher air velocity. The latter seems to increase the energy output of the turbine in this type of configuration. From various results obtained in the CFD investigation by Abe (2004), it became clear the optimum loading coefficient for every variation of the diffusers angle, length and brim height needs to be determined. The CFD investigation was followed with wind tunnel experiments by Abe et al. (2005). The Brim (200mm) was substantially larger than the brim of Matsushima et al. (2005) and the diffuser angle was also increased. The wind turbine used in the experiment had a diameter of 400mm. Figure 2.12 prove that the power coefficient of the wind turbine with the diffuser was substantially higher than the open wind turbine. The energy output is much higher than the diffuser with brim of Matsushima et al. (2005), this can be dedicated to the larger diffuser angle and brim. From the investigation it was also noticed that the shrouded wind turbine's peak performance was at a higher tip speed ratio than that of the open wind turbine. Figure 2.12 also shows that the experimental data and CFD modelled power coefficient results correspond well.

Ohya et al. (2008) did some experiments with different configurations of components. The results are presented in Figure 2.13. The total length divided by the inlet diameter is denoted

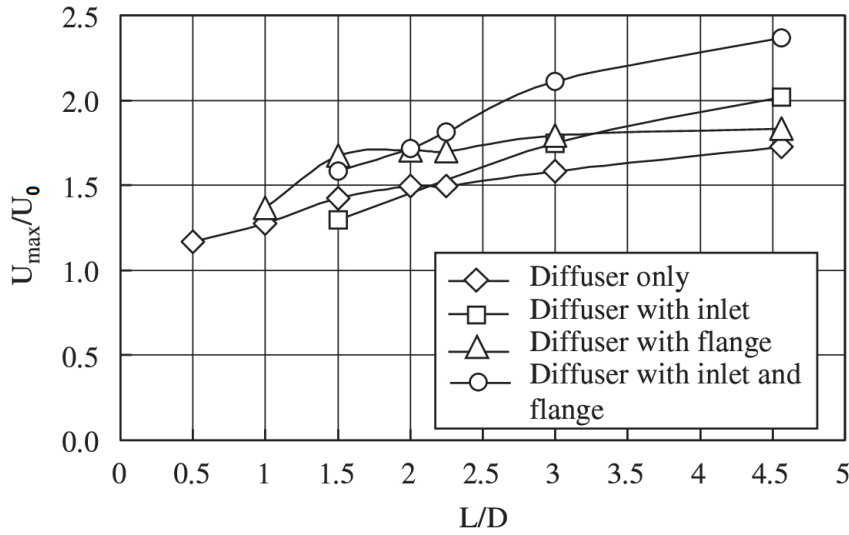


Figure 2.13: Velocity increase with different configurations of components and length ratio's (Ohya et al. 2008)

on the horizontal axis of the graph. The free wind speed is U_0 and U_{max} is the maximum air speed as measured at the throat. It is evident that the configuration with diffuser, brim and inlet shroud offers the best velocity ratio. Another important feature to be taken into account, is the length of such a configuration. Results proved that the air speed inside increases when the diffuser is lengthened. However, caution to apply a very long structure is emphasized as it will have practicable constraints, for example when to be constructed on a tower. For the field test an 8m tower was erected by Ohya et al. (2008) and a diffuser inlet diameter of 0.72m was decided upon with a total length of 0.9m and brim height of 0.36m was applied. The practical and calculated results show a $C_p = 1.4$ compared to a $C_p = 0.35$ for the open turbine. Ohya et al. (2008) also developed and built a number of compact shrouded wind turbines with the same configuration as above with total length divided by the inlet diameter of 0.22 and a total diameter of 2.5m. The wind turbines were rated as 5 kW. A $C_p = 0.54$ was obtained when the total outer diameter (brim included) was used to calculate the power coefficient. This is yet an exceptional performance as most wind turbines on the market only have a power coefficient of $C_p = 0.4$.

From the field devices that were tested it could be seen that the C_p value of a DAWT was greater. This is also the case even though the outer diameter of the flange is used as reference and not the blade maximum diameter. The numerical investigation by Abe (2004) and wind tunnel experimental results by Abe et al. (2005) also revealed the advantages of a shrouded wind turbine compared to an open wind turbine. This configuration could thus be used to improve the extraction of energy from low speed wind for more efficient power generation.

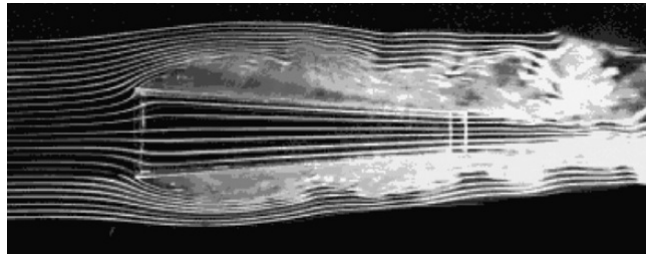


Figure 2.14: Concentrator in a wind tunnel (Ohya et al. 2008)

2.5 Concentrators

Concentrated wind will increase the power yield in relation with the rotor-swept area (Hau 2006). A wind turbine in the concentrator will encounter a higher air speed and rotate at a higher revolution per minute. This wind turbine will also start rotating at a lower free wind speed as the concentrator amplifies the air speed. Therefore, the concept of a concentrator should be beneficial. Recent work, as will be investigated further in this chapter, will give some insight into this concept.

The experimental work with concentrators by Shikha et al. (2005) found that a concentrator with an outlet to inlet ratio of 0.15 displays the best increase of 4 to 4.5 times the free wind speed, at the outlet. If the increase was calculated with continuity and incompressibility (at low Reynolds numbers) the speed at the outlet should have been 6.7 times that of the free wind speed. From this it can be concluded that some of the mass flow tends to avoid the concentrator. This is a result of the sudden increase in area at the outlet of the concentrator, skin friction drag and pressure drag. These losses create a resistance to flow while a free wind stream usually evades such obstacles. Ohya et al. (2008) also experimented with concentrators and the outcome is depicted in Figure 2.14 which confirms the results of (Shikha et al. 2005). Ohya et al. (2008) concluded that the wind tends to avoid the nozzle-type model.

Recently, concentrators are mainly used in configurations with vertical axis wind turbines. The air flow is concentrated and deflected away from the one side of the horizontal blades, thus reducing drag and increasing the power output of the wind turbine (Orosa et al. 2009).

However, for HWAT's there is only new developments with a concentrator in conjunction with a diffuser (Figure 2.15). As the diffuser is fixed to the outlet of the concentrator, the losses of energy that occur with the sudden increase in area are eliminated. Wang et al. (2007) recently did CFD simulations and wind tunnel tests on a concentrator with diffuser configuration. When the wind turbine was fitted into the shroud the captured energy increased with 43% for the same free wind speed. This emphasizes the importance of a shrouded wind turbine. It is further proposed that the configuration should rather be build-in or mounted on a structure than mounted on a pole.

Wang et al. (2008c) concluded after extensive research that an existing wind turbine cannot

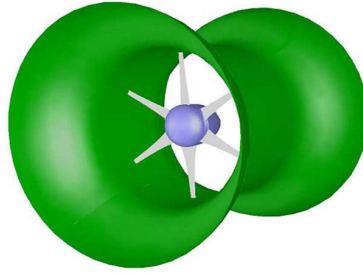


Figure 2.15: Concentrator, Diffuser type of wind turbine (Wang et al. 2007)

be used in the shroud. The need was confirmed for newly designed blades and hub to suite the conditions in the shroud.

2.6 Computational fluid dynamics (CFD)

The increase in computational power and enhanced software has lead to CFD to become a useful tool for researchers to develop new concepts and optimize it. This section focuses on the research that has been done by implementing CFD to simulate shrouded diffusers.

Wang et al. (2008c) investigated a concentrator and diffuser arrangement for a shrouded wind turbine. The error between the wind tunnel tests and the CFD was within 5% and the results also indicated that the design will improve energy capture at lower wind speeds. The CFD results were validated with the measurement results of the wind tunnel tests and at the time of writing the article, they were using CFD to design a hub and test some different types of blades configurations. They used the $k - \varepsilon$ model, incompressible flow and steady state flow field to simulate the arrangement with velocity inlet and pressure outlet. Blockage can be a factor if the domain around the model is too small, they extended the domain in the axial direction 4.5 times the diameter of the rotor and in the cross section 3.6 times the diameter. Half of the test domain was modelled with the use of a symmetry plane. Fine grids were used on the blades, the diffuser and concentrator, with three boundary layers to more accurately predict the pressure and viscous forces. The remain of the domain had a grid with Tetrahedral elements. The total number of elements was 150000 for the total length and diameter of 0.92m for the diffuser.

Wang & Chen (2008) investigated the effect of blade numbers in shrouded diffusers and the angle of attack on power production. They also used the $k - \varepsilon$ model with a pressure outlet and velocity inlet. The results indicated a great variance in the output with the chance of blade numbers and angle of attack. It showed that CFD is a powerful design tool to determine a good configuration for the variables at hand.

2.7 Summary and proposed configuration

2.7.1 Summary

As evident from the momentum theories, performance (C_p) is less at a lower local blade speed ratio at the root. Therefore, it could be beneficial if a root section of the blade is sacrificed to increase the air velocity to the tip region of the blade.

As the axial concentrator diffuser arrangement is quite lengthy, it is not desirable to apply to a pole mounted device. It will be a better alternative, if possible, to have a compact concentrator diffuser arrangement, with yaw control that can be mounted on a tower.

The literature study on diffusers proves the necessity for a shroud design with an inlet, outlet shroud and flange. A new wind turbine in the shroud should be designed, accordance to the local conditions in the shroud. From the CFD investigation of Abe (2004) it became clear that the loading coefficient has to be low. It could therefore be concluded that a smaller blade that rotates at a higher revolution and lower torque will be beneficial for energy output. As the amount of torque exerted on the turbine blades will not be high it would also imply a smaller cord length, also beneficial for solidity.

Some other advantages for this type of configuration include:

- An increase of power output if compared to conventional wind turbines.
- The fact that the flow over the tip of the blades can be reduced with the shroud, can increase efficiency.
- The possibility to significantly reduces aerodynamic noise makes it a favourable choice for urban locations (Ohya & Karasudani 2010).
- Safety is improved as the wind turbine rotates in a shroud (Ohya & Karasudani 2010).
- Depending on the height of the brim, it provides a degree of yaw control (Ohya & Karasudani 2010).
- As the wind turbine will rotate at a higher speed it will also use a smaller and less expensive generator (Wang et al. 2007) that needs lower torque for starting.

2.7.2 Proposed configuration

After considering the literature it was decided to have a larger center hub region that can be used to concentrate the air flow to the wall inside the diffuser to increase the air velocity in this annular area between the diffuser inner wall and the new hub. The turbine can then be situated in this annular area. The new root section is then a increased distance away from the center axis and

should therefore have a higher local speed ratio with increased performance. A much lesser blade solidity at the new root area should also improve the performance in that region. This creates a compact arrangement that can still be pole mounted. As seen from the back of the diffuser, it should be the same as a conical type (Figure 2.10) with a brim as depicted in Figure 2.11. The hub can then be an airfoil shape revolved around the center axis of the diffuser, to reduce the amount of drag to keep the internal flow to a maximum. The amplified air speed at the turbine should contribute to the starting of the blades to compensate for the loss of a lower blade region.

Existing blade element theories can be used to design a blade for the new shrouded wind turbine. The optimum blade angle can then be determined with the use of CFD for the new configuration.

Chapter 3

Theoretical background and CFD simulation setup

3.1 Introduction

The Betz limit (Betz 1926) has been derived to introduce a theoretical limit to the energy that a wind turbine can subtract from a free wind stream. This limit shows how efficient current and newly developed wind turbine are. The equations necessary to design a basic blade for the shrouded wind turbine is set out in the next section. As CFD was used to determine the power output of the open and shrouded wind turbines, the setup in Star-CCM (Program used to model the wind turbines) and boundary conditions are elaborated upon at the end of this chapter.

3.2 Available power

The maximum theoretical power which can be extracted from the wind is set out below. This law is derived from the principles of conservation of mass and momentum and is generally attributed to Betz (1926), although there was three independent discoveries. The following assumptions are made in order to derive the maximum power available.

- Homogeneous, incompressible, steady state fluid flow
- No frictional drag
- An infinite number of blades
- Non rotating wake
- Uniform thrust over the rotor area
- Static pressure far upstream and downstream is equal

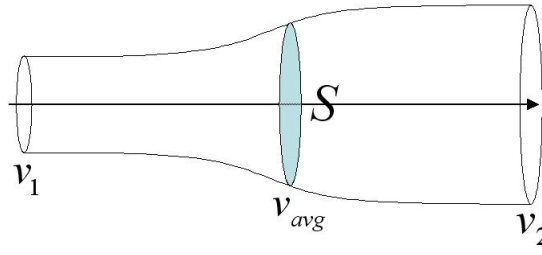


Figure 3.1: Actuator disk model for a wind turbine

Figure 3.1 reveals the control volume used to derive the limit. Equation(3.1) shows conservation of mass in the stream-tube.

$$\dot{m} = \rho \cdot A_1 \cdot v_1 = \rho \cdot S \cdot v = \rho \cdot A_2 \cdot v_2 \quad (3.1)$$

Here v_1 is the speed in the front of the rotor, v_2 is the speed downstream of the rotor and the speed at the disc is v . The fluid density is ρ and the area of the turbine is given by S . The force exerted on the wind by the rotor:

$$F = m \cdot a \quad (3.2)$$

$$= m \cdot \frac{dv}{dt} \quad (3.3)$$

$$= \dot{m} \cdot \Delta v \quad (3.4)$$

$$= \rho \cdot S \cdot v \cdot (v_1 - v_2) \quad (3.5)$$

The work done by the force.

$$dE = F \cdot dx \quad (3.6)$$

The power of the wind is

$$P = \frac{dE}{dt} = F \cdot \frac{dx}{dt} = F \cdot v \quad (3.7)$$

Substituting the force into the power equation will yield the power extracted from the wind:

$$P = \rho \cdot S \cdot v^2 \cdot (v_1 - v_2) \quad (3.8)$$

Power can also be computed by using the kinetic energy.

$$P = \frac{\Delta E}{\Delta t} \quad (3.9)$$

$$= \frac{1}{2} \cdot \dot{m} \cdot (v_1^2 - v_2^2) \quad (3.10)$$

With (3.1) it yields the following

$$P = \frac{1}{2} \cdot \rho \cdot S \cdot v \cdot (v_1^2 - v_2^2) \quad (3.11)$$

Equating the two power expressions yields

$$P = \frac{1}{2} \cdot \rho \cdot S \cdot v \cdot (v_1^2 - v_2^2) = \rho \cdot S \cdot v^2 \cdot (v_1 - v_2) \quad (3.12)$$

That gives

$$\frac{1}{2} \cdot (v_1^2 - v_2^2) = \frac{1}{2} \cdot (v_1 - v_2) \cdot (v_1 + v_2) = v \cdot (v_1 - v_2) \quad (3.13)$$

or

$$v = \frac{1}{2} \cdot (v_1 + v_2) \quad (3.14)$$

Returning to the previous expression for power based on kinetic energy and substituting (3.14)

$$\dot{E} = \frac{1}{2} \cdot \dot{m} \cdot (v_1^2 - v_2^2) \quad (3.15)$$

$$= \frac{1}{2} \cdot \rho \cdot S \cdot v \cdot (v_1^2 - v_2^2) \quad (3.16)$$

$$= \frac{1}{4} \cdot \rho \cdot S \cdot (v_1 + v_2) \cdot (v_1^2 - v_2^2) \quad (3.17)$$

$$= \frac{1}{4} \cdot \rho \cdot S \cdot v_1^3 \cdot \left(1 - \left(\frac{v_2}{v_1} \right)^2 + \left(\frac{v_2}{v_1} \right) - \left(\frac{v_2}{v_1} \right)^3 \right) \quad (3.18)$$

By differentiating $\dot{E}$ with respect to $\frac{v_2}{v_1}$ one finds the maximum or minimum value for $\dot{E}$. The result is that $\dot{E}$ reaches a maximum value when $\frac{v_2}{v_1} = \frac{1}{3}$. Substituting this value results in:

$$P_{\max} = \frac{16}{27} \cdot \frac{1}{2} \cdot \rho \cdot S \cdot v_1^3 \quad (3.19)$$

The obtainable power from a cylinder of fluid with cross sectional area S and velocity v_1 is:

$$P = C_p \cdot \frac{1}{2} \cdot \rho \cdot S \cdot v_1^3 \quad (3.20)$$

The total power is

$$P_w = \frac{1}{2} \cdot \rho \cdot S \cdot v_1^3 \quad (3.21)$$

The power coefficient

$$C_p = \frac{P}{P_w} \quad (3.22)$$

has a maximum value of: $C_p = 16/27 = 0.593$

3.3 Total power available in a shroud

Equation 3.21 is generally accepted to determine the total power available in a diffuser or concentrator as proposed by Orosa et al. (2009), Wang et al. (2007), Bernard Frankovic' & Vrsalovic (2001) and Ohya & Karasudani (2010) (naming only a few). The average velocity where the wind turbine should be situated in the shroud is measured and substituted in the place of v_1 in equation (3.21) to determine the total power available. The total blade area of the turbine in the shroud is denoted as S .

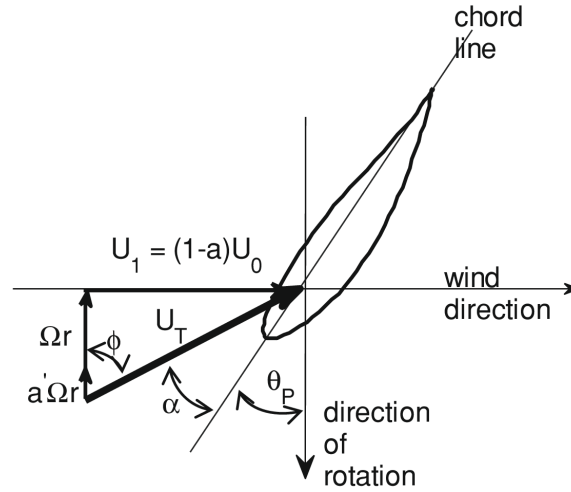


Figure 3.2: Velocities for a cross-section blade element at radius r (Wood 2011, 45).

3.4 Blade design

The variables as depicted in Figure 3.2 and elaborated on in Chapter 2 can be used to determine the torque per blade element. U_T is the relative velocity. U_0 is the axial velocity, a the axial induction factor, a' the rotational induction factor, r the radius at the centre of the blade element and Ω in rad/s .

The torque (dQ) available for a blade element from the velocity of the air, for the annular area (dA) can be determined with (3.25). This equation is derived from (3.23) and (3.24). N is the number of blades of the wind turbine.

$$dP = \Omega \cdot dQ \quad (3.23)$$

$$dP = C_p \cdot \rho \cdot U_0^3 \cdot dA \quad (3.24)$$

$$dQ = \frac{C_p \cdot \rho \cdot U_0^3 \cdot dA}{\Omega \cdot N} \quad (3.25)$$

The angle ϕ in Figure 3.2 can be determined with (3.26) (Wood 2011, 47).

$$\tan \phi = \frac{U_0 \cdot (1 - a)}{\Omega \cdot r \cdot (1 + a')} \quad (3.26)$$

The torque per blade element (Wood 2011, 46) can also be determined with the drag and lift forces on the blade element represented with (3.27). The two dimensional lift (C_l) and drag (C_d) ratios should be determined for the airfoil after the Re is determined with c (chord length) and U_T .

$$dQ = 0.5 \cdot \rho \cdot U_T^2 \cdot c \cdot (C_l \cdot \sin \phi - C_d \cdot \cos \phi) \cdot r \cdot dr \quad (3.27)$$

Equation (3.27) should be equal to (3.25) for each blade element. Therefore designing each blade element, requires an iterative process.

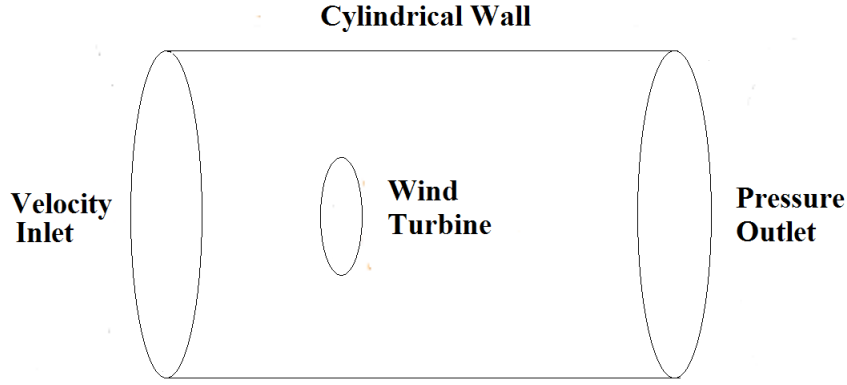


Figure 3.3: Boundaries for the simulation domain in CFD

3.5 CFD simulation set-up

The simulations that were done with the Star-CCM+ software necessitate an elaboration on the set-up of the wind turbine models in this CFD code. Section 2.6 on page 19 dealt with simulations of shrouded wind turbines in CFD and explained the way in which wind turbines in shrouds were simulated in a CFD Code. This was used as a basis to set-up the wind turbine configurations in CFD. The set-up explained in this section was used in the validation of the computational modelling as well as the simulations to design the shrouded wind turbine.

A three dimensional CAD package Solidworks was used to draw the wind turbine configurations. The drawings were then imported into Star-CCM+ as a surface mesh. The whole domain was volume meshed with a polyhedral mesh. On the surfaces of the diffuser and blades a prism-layer mesh was used to mesh the boundary layer. The three dimensional flow field was simulated as steady state. A uniform velocity inlet and pressure outlet was chosen as inlet and outlet boundaries. As the wind turbine configurations are cylindrical, the wall boundary was also drawn cylindrically. As a boundary layer was not necessary, the shear stress specifications were chosen as slip on these boundaries. This domain can be seen in Figure 3.3.

In some simulations, a chosen angle of this cylindrical domain was simulated to reduce computing time. Periodic interfaces (Wang & Chen 2008) were used to model some of the shrouds with blades. For six blades only 60° of the cylindrical domain was modelled and for three blades 120° . Symmetrical boundary conditions were applied when an angle of the domain was modelled for only the shroud (without a wind turbine).

The low Mach numbers lead to the use of constant density (incompressible flow) to model wind turbines either with or without a shroud as well as the shrouds with no wind turbine (Wang et al. 2008a).

The flow fields were defined with the Reynolds-Averaged Navier-Stokes (RANS) equations. The equations were completed with the use of additional turbulent models. This additional transport equations that were solved along with the RANS flow equations was the $k - \epsilon$ turbulence or $k - \omega$ turbulence models (Versteeg & Malalasekera 2007, 66).

The two layer $k - \epsilon$ model with standard wall function was used to obtain cell independence, but near-wall performance is unsatisfactory. Thus for increase accuracy a $k - \omega$ model with a Gamma RTheta transition model (Langtry 2006) was introduced after cell independence was reached. The model was implemented with a field function that defines the free stream edge. The $k - \omega$ model required more computing resources, therefore cell independence was initially reached with the two layer $k - \epsilon$ model.

The Star (2014) help file proposed a segregated flow model to solve the incompressible flow, which also saved computing costs.

3.6 Summary

In this chapter the available power in the free wind was derived and the necessary blade element theory equations was set out. The set-up for the CFD simulation program was elaborated upon.

Chapter 4

Validation

4.1 Introduction

This chapter shows the process that was followed to validate the computational modelling of the wind turbine configurations in CFD. The two primary components (Figure 4.1) of the new design are the shroud with brim and the wind turbine. The computational results obtained were compared with experimental data (Oberkampf & Trucano 2008).

The CDF set-up as described in Chapter 3.5 was implemented to model the shroud with brim and an open three bladed wind turbine.

4.2 Criteria for meaningful CFD results

In order for a simulation to generate results that are meaningful, requires primarily that a value of significance converge from a number of iterations and that cell independence is maintained (Versteeg & Malalasekera 2007, 5). This was accomplished by plotting these values (velocity at a point in the shroud and torque for the wind turbine blades) against iterations. Each time a surface mesh changed, this value (the solver) should converge. If these values converged, the surface mesh size was reduced and the model was again simulated until the same values converged again. If this process is followed and the converged values remain the same, cell independence is reached.

Another value of importance is the Wall Y^+ value that indicates if the boundary layer was sufficiently discretized with prism-layer cells. Wang & Chen (2008) indicated that the Wall Y^+ value should be in the range of one and zero to solve the laminar sub-layer accurately.

Residuals are produced after each one of the iterations. This indicates how well the governing equations are numerically satisfied for each solver. According to the (Stern et al. 1999), a value below 0.001 is more likely for complex geometry and conditions than values nearer to 0.

As the measured values and simulated values were different it was necessary to define tolerances for these differences. Babuska & Oden (2004) proposed in their study on validation and verification

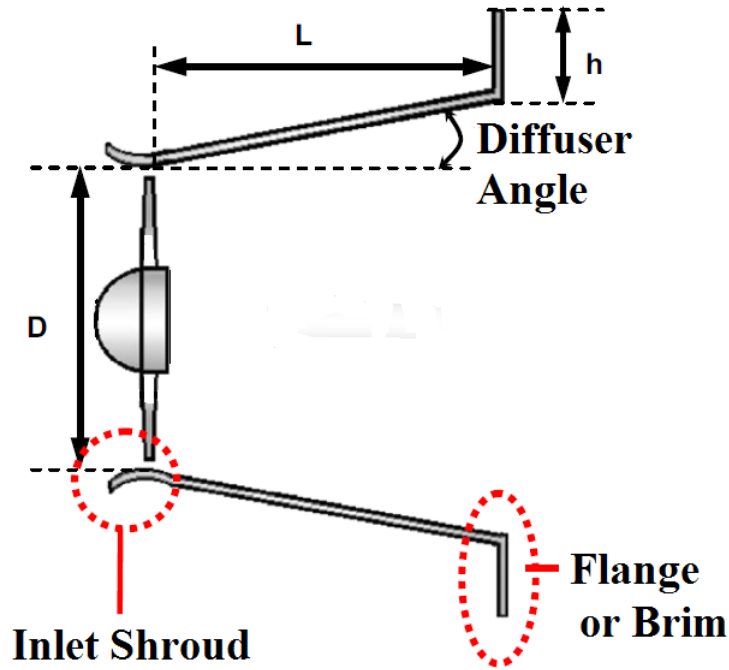


Figure 4.1: Schematic of a wind turbine equipped with a flanged diffuser shroud (Ohya et al. 2008)

in computational engineering and science that these tolerances are user defined and will vary with the purpose of the values obtained. In the study on a wind turbine in a shroud Wang et al. (2008b) showed that a 5% difference existed between the measured and simulated results.

4.3 Case 1: Diffuser

4.3.1 Diffuser parameters

Ohya et al. (2008) performed wind tunnel experiments on diffusers with a brim attached to the outlet. The wind tunnel velocity U_0 was $5m/s$. The length(L), inlet diameter(D) and brim length(h) is depicted in Figure 4.1. The inlet diameter of the diffuser was $D = 20cm$ and the ratio to obtain the length was $\frac{L}{D} = 1.5$. The area ratio was 1.44 for the inlet and outlet diffuser surface area. The velocity in the diffuser was measured at the central axis with a I-type hot wire and a static-pressure tube. The size of the brim was varied to obtain a optimum height for a maximum velocity at the inlet of the diffuser. The height of the brim is given as a ratio $\frac{h}{D}$. Figure 4.2 illustrates the values obtained through the experiment. For the values obtained by Ohya et al. (2008) the validation is done on the diffuser with a brim that had a height of $\frac{h}{D} = 0.25$.

4.3.2 Diffuser simulation set-up

To reduce the computing time only half of the domain was modelled. Therefore a symmetry plane was selected for the plane that cut the domain in half. As the wall on the cylindrical boundary

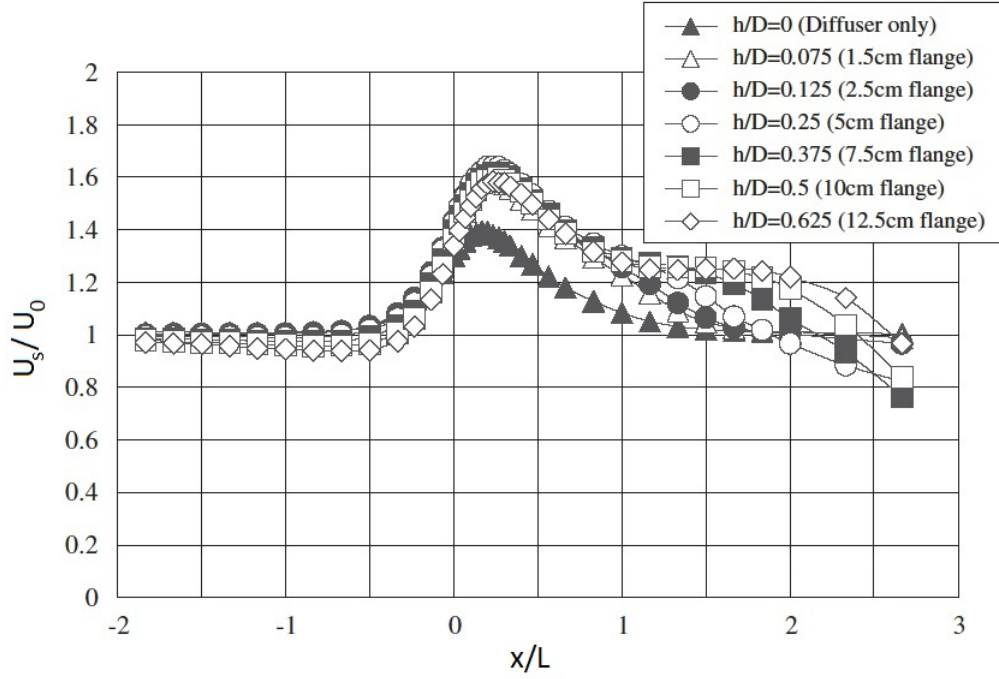


Figure 4.2: Wind velocity distribution on the central axis of a circular-diffuser with different brim heights (Ohya et al. 2008).

should not have a boundary layer the prism-layers were disabled and the shear stress specifications were chosen as slip. The domain (Figure 4.3) was drawn in Solidworks in such a manner that the boundaries do not influence the velocity at the diffuser (Wang & Chen (2008)). A large surface mesh was chosen for the boundaries and smaller ones for the shroud and volume at the back of the shroud. The prism-layer was disabled for the small areas that represent the thickness of the shroud wall. The velocity inlet value was set at 5 m/s , the same as the wind tunnel velocity.

Criteria of Chapter 3.5 completed this set-up proses.

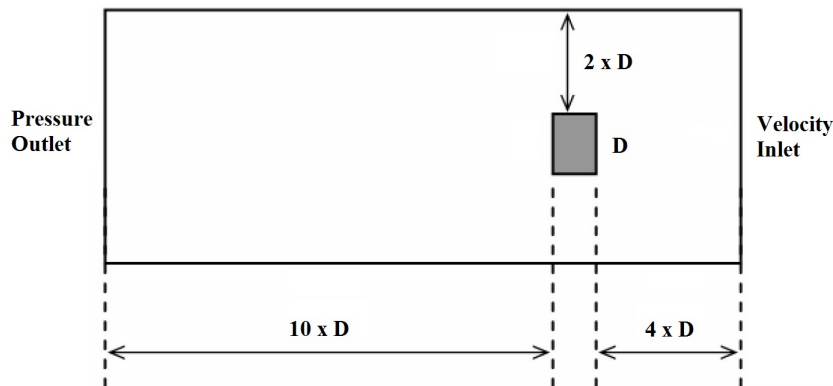


Figure 4.3: Simulation domain for the diffuser without a wind turbine

4.3.3 Proses followed to reach cell independence for the diffuser simulations

For the walls of the diffuser the mesh size was chosen as $2mm$ for the relative minimum size and the target size as $6mm$. For the shroud inner and outer walls 15 prism-layers were chosen with a total height of $7mm$ to simulate the boundary layer. The mesh values for the velocity inlet pressure outlet and symmetry plane were chosen as $250mm$ for the relative minimum size and the target size is $400mm$. A volumetric cylinder was inserted at the back of the shroud in the area of low pressure and high turbulence (Figure 4.4). This size was chosen as $20mm$.

The point of significance was chosen on the center axis of the shroud near the region of maximum velocity. The velocity (magnitude) at this point was plotted (monitor plot) against iterations to show the influence of changing the mesh and therefore the number of cells in the domain.

The surface mesh on the boundaries was reduced in size until it had no influence on the velocity (magnitude) for the selected point in the shroud. The mesh values at the inlet, outlet symmetry plane and wall of the model's relative size were reduced from $250mm$ to $200mm$. The target size was reduced from $400mm$ to $350mm$. The point in the shrouds velocity changed from $7.58m/s$ to $7.57m/s$. It was thought that the mesh size was not significantly altered the minimum size was reduced to $50mm$ and the relative size to $100mm$. This also did not have a significant effect as previously. Thus the surface mesh was set to the original chosen size prior to the optimisation, thus $250mm$ for the relative size and $400mm$ for the target size.

As the change in surface mesh size of the outer boundaries did not have a significant effect, the original program that converged at 8180 (Figure A.1) iterations was used to simulate the variation in mesh size in the volume at the back of the diffuser (densely volume meshed region as depicted in Figure A.3). The mesh value for the volume was reduced from $20mm$ to $10mm$. After meshing the domain the total cells were 664350. The monitor plots value changes significantly from $7.58m/s$ to $7.28m/s$ after it converged (Figure A.1). As this had a great influence, the volume mesh was once again halved to $5mm$ at 13290 iterations. The total cells increased to 3.5 million and the velocity chanced from $7.28m/s$ to $7.195m/s$. The volume mesh was then chanced to $3mm$. This did not effect the plot and cell independence was thus reached for this area.

Returning to the simulation with the volume mesh of $5mm$, the mesh values of the diffuser and brim were altered to half the original value. The thickness of the prism-layers was also changed to $3.5mm$ from $7mm$. This changed the value of the point in the diffuser from $7.19m/s$ to $7.16m/s$ (0.4%). As this does not represent a significant value, cell independence was reached for the diffuser simulation.

A $k - \omega$ model with a Gamma R ϵ theta transition model (Langtry 2006) was then adopted. The model was implemented with a field function that defines the free stream edge as $5mm$. This increase the value of the point in the diffuser to $7.36m/s$ as illustrated in Figure A.1. After the

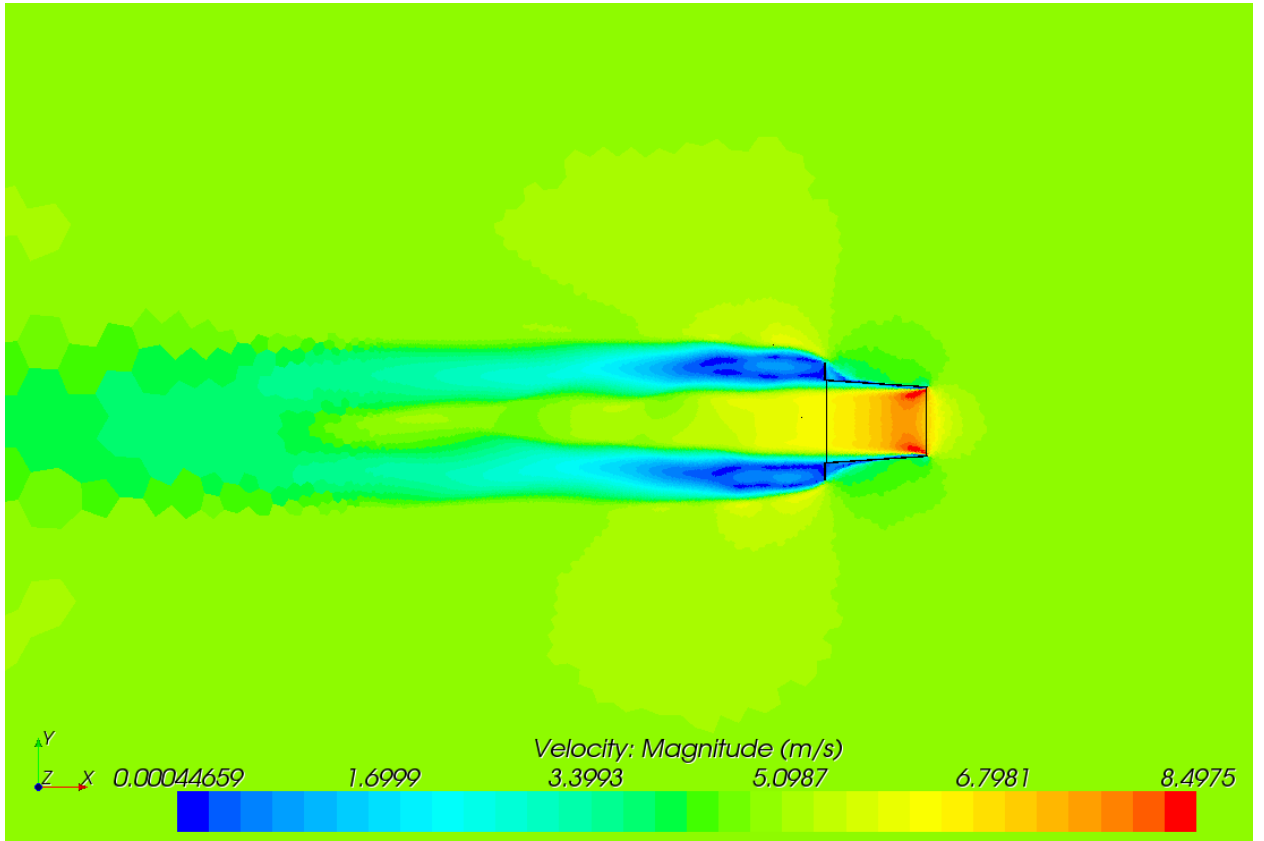


Figure 4.4: Velocity(magnitude) for the diffuser without wind turbine

new model was implemented the velocity had a small variation between $7.3353m/s$ and $7.38m/s$. The implementation of the new model significant increased the predicted velocity in the shroud and can be attributed to the influence of the boundary layer in a diffuser.

4.3.4 Reflection on results and CFD solve information of the diffuser simulation

The velocity (magnitude) for the domain is illustrated in Figure 4.4. This confirms that the boundaries have no influence on the values of the velocity in the diffuser. As it was suspected that the outlet boundary was still in an area where the flow was not normalized, it was extended. It did not have any effect on the velocity in the diffuser. This increased the computing requirements, and the original domain was restored as suggested by Wang & Chen (2008).

The volume mesh for the cell independence simulation can be seen in Figure A.3. It is evident that the cells in the low velocity region at the back of the diffuser are as densely packed as the cells in the diffuser. This greatly increased the computing time. The prism-layers volume mesh to model the boundary layer of the diffuser is set out in Figure A.4.

The Wall Y^+ value for the boundary layer inside the diffuser is shown in Figure 4.5. The figure points out this value for the inner wall of the diffuser. For a small area, with the highest velocity, the value was 1.25. The rest of the Wall Y^+ values were less than 1. The residuals are

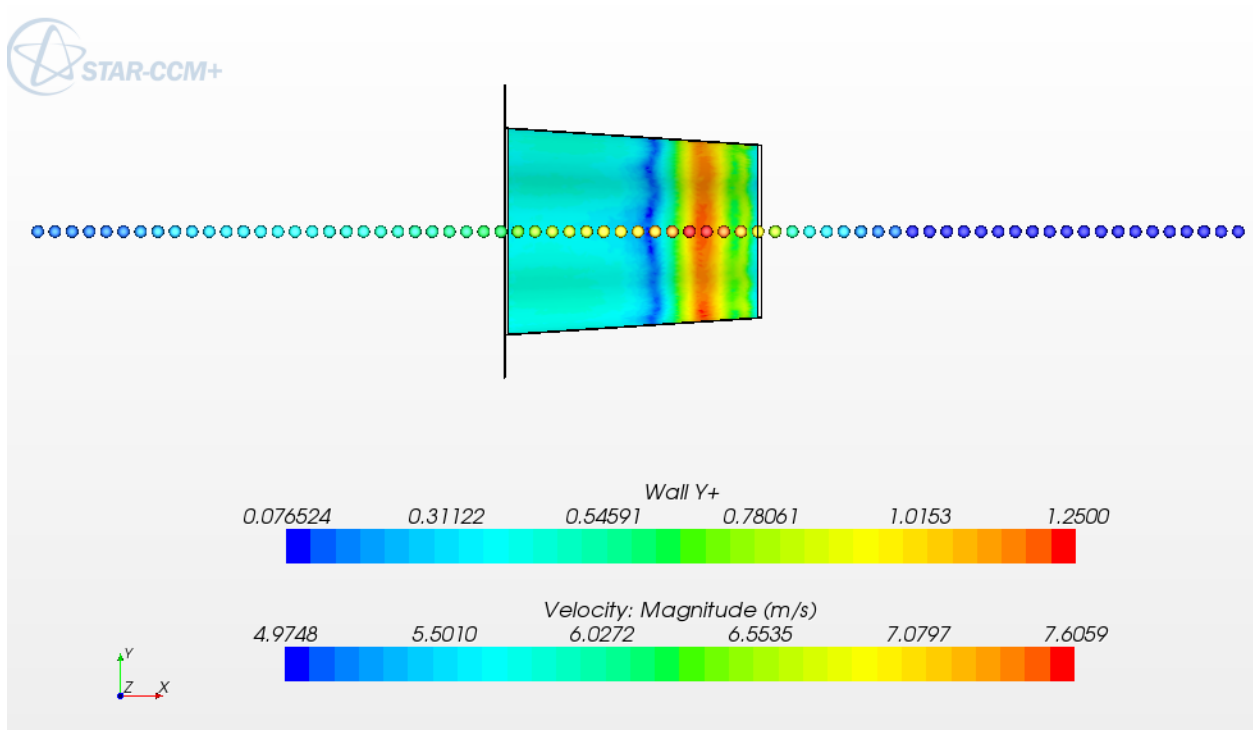


Figure 4.5: Wind velocity(magnitude) distribution on the central axis of the diffuser and Wall Y+ values

demonstrated in Figure A.2 and was satisfactory.

The velocity (magnitude) on the central axis of the diffuser for the cell independent simulation, with transition model is shown in Figure 4.5. These values were divided with the inlet velocity ($5m/s$) to present a ratio for the y axis and is plotted in Figure 4.6 with the measured values. The x axis is $(\frac{x}{L})$ with L the length of the diffuser and 0 is the inlet of the diffuser. The diffuser is thus situated between 0 and 1. The simulation results compare well to the measured values and the trend is followed from well in advance of the diffuser ($\frac{x}{L} = -2$) to a distance ($\frac{x}{L} = 4/3$) at the back. The values further away do differ but is not a concern as the wind turbine is situated in the diffuser. The wind turbine will naturally be situated at the point of maximum velocity (U_s) in the shroud and this value will be the peak value on the plot. The simulation predicts a value of $\frac{U_s}{U_0} = 1.52$ and the measured value was $\frac{U_s}{U_0} = 1.64$ with a difference of 8%.

4.3.5 Summary of the diffuser simulations

Measured values have some degree of inaccuracy. Ohya et al. (2008) also had to remove the walls and the ceiling to reduce the blockage effect in their wind tunnel. This shows that the experimental values could be compromised as the wind tunnel itself was altered.

In the simulation process it was found that an increase in the number of cells in the domain results in an increase in wind velocity values up to a certain point in value where after it starts

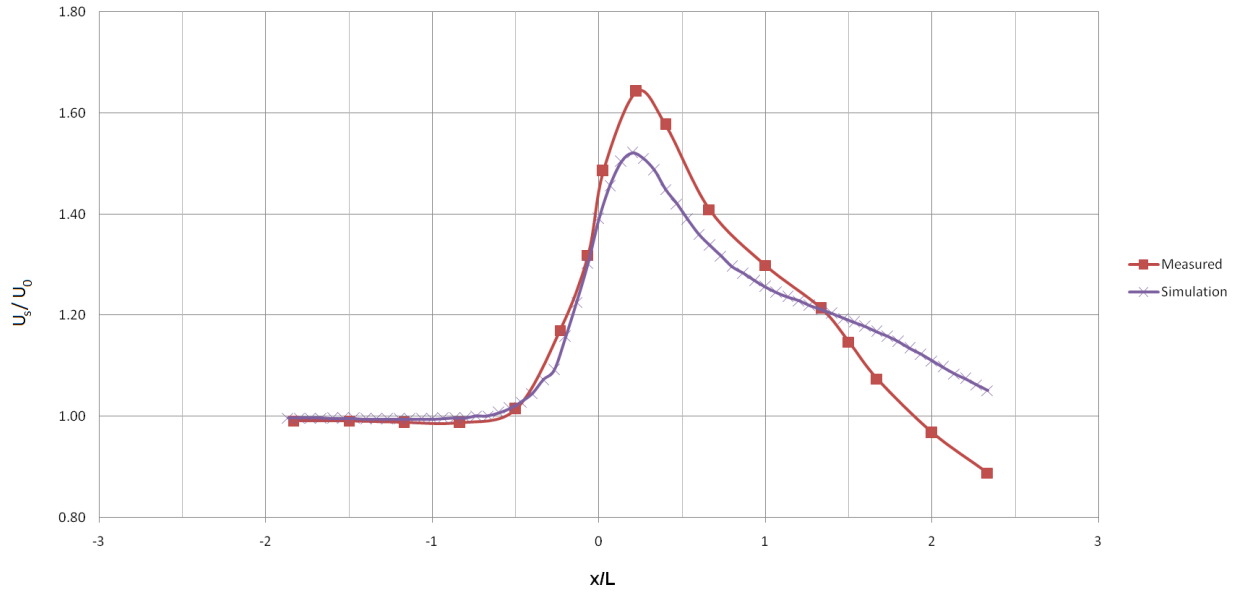


Figure 4.6: Air velocity(magnitude) plot on the central axis of the diffuser model in CFD and measured values

to decrease until cell independence is reached. It was also found that the increase in prism-layers on the wall of the diffuser also decreased the velocity until cell independence is reached. This was tested in several simulation beforehand.

Wang et al. (2008b) had a 5% difference in error with three boundary layers on the turbine and scoop. Seven boundary layers were used to obtain the results as set out in the simulations as described in this section. This indicates that the value could be higher than the 5% that Wang et al. (2008b) previous obtained. It was found that an increase of the number of prism-layer on the boundary did indeed influence values. According to this, an 8% difference would not be significant. It is also encouraging that the simulation values do not over predict the velocity in the diffuser as this would over predict the performance of a wind turbine in a diffuser. Therefore a conservative value will be obtained in a design simulation with a turbine and diffuser. It can thus be concluded that the simulations provided a sufficient reflection of the reality.

4.4 Case 2: Open wind turbine

4.4.1 Wind turbine parameters

A three bladed 3.6m wind turbine (*AE 1.0kW*) was developed by Aero energy for low wind speed regions like South Africa. Measured results, CAD drawings and predicted results were obtained for this wind turbine (Bosman et al. 2003). These values were used to validate the simulation of a wind turbine in CFD. Thus, the measured power to wind speed curve (Figure 4.10) was compared

Velocity (m/s)	Ω (rad/s)
3	12.57
4	16.12
5	17.88
6	18.80
7	19.58
8	20.41
9	21.47

Table 4.1: Predicted rotational speeds of the *AE 1.0kW* wind turbine at certain wind speeds

to the modelled results.

The values in Table 4.1 was used to simulate the wind turbine. The predicted rotational value was inserted in the CFD program to simulate the wind turbine at each wind speed, as measured RPM values were not available. These values in Table 4.1 were used instead of simulating the wind turbine at each wind speed a number of times at different rotational speeds to obtain a power to tip-speed curve. This would have increased the number of simulations substantially.

4.4.2 Wind turbine simulation set-up

The moment (torque) of the blade was chosen as a value of significance to show if convergence and cell independence was reached. The wind speed (inlet boundary) and predicted rotational speed were entered in the simulation to produce a moment on the rotational axis. From the measured results for a wind speed of $8m/s$ a moment of $50Nm$ was obtained for the *AE 1.0kW* wind turbine at $21.99rad/s$. If the modelling of the wind turbine at a inlet air speed of $8m/s$ produces a torque value near $50Nm$ after cell independence is reached, the moment's on the blade can be obtained for different wind speeds and compared with the measured values.

The simulation of a large domain and a rotating wind turbine required a reasonable amount of computing power. A basic domain around the wind turbine was chosen and then with the use of extruded meshes was extended to determine the influence on the moment. This assisted in minimising the number of cells. The chosen domain is illustrated in Figure 4.7. To evaluate influence of the boundaries the inlet was extended to $1.7m$ from the wind turbine, the outlet to $12.2m$ and the cylindrical boundary's radius was increased $1.6m$ separately in the proses to reach cell independence, to have the same domain as proposed by Wang & Chen (2008). The inlet and outlet extruded meshes did not influence the torque, but the increase of the cylindrical boundary was necessary as seen in Figure 4.8.

The surface mesh of the domain and three bladed wind turbine was imported into the simulation.

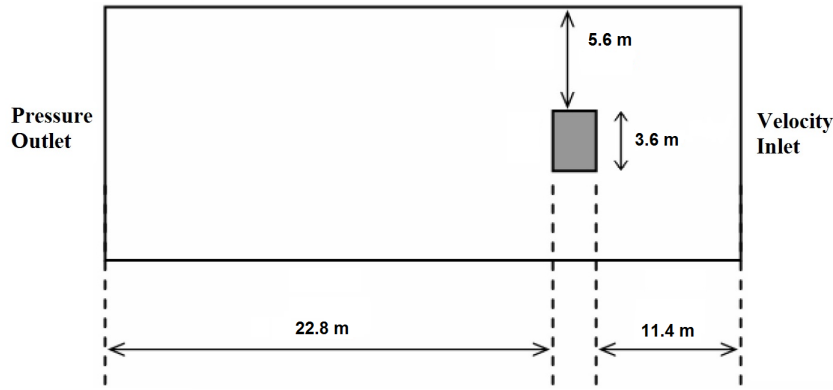


Figure 4.7: Domain for the simulation of the open wind turbine

A small region around the blades, Figure 4.9 in yellow, was connected with the large domain with in place interfaces. As the wall on the outer boundary should not have a boundary layer the prism-layers was disabled and the shear stress specifications was chosen as slip.

Creating a monitor plot of the torque on the blade a rotational axis in the centre of the blade was chosen perpendicular to the radius of the blade. The right hand rule was use to specify the direction of rotation. Rotating reference frames was chosen to rotate the blades.

The near core layer aspect ratio was chosen as 2 as Star-CCM+ has a maximum recommended value of 2.5.

Criteria of Chapter 3.5 completed this set-up proses.

4.4.3 Proses followed to reach cell independence for the wind turbine simulations

After the appropriate models were selected, boundaries specified and interfaces created the cell size of the surface mesh had to be specified. A base size base size of $500mm$ for all cells with a relative minimum size of 25% of the base and a target size of 100% of base was applied. The boundary layer was enabled only on the surface of the blade. Two boundary layers 33% of the base were chosen. Figure 4.9 shows the surface mesh that was created. The positive direction of the axis was chosen with the right hand rule. Simulating this without rotating the blade gave a positive torque of $7.8Nm$ that indicated that the direction of rotation is correctly chosen. Rotating the blade at the required rotational speed of $21.99rad/s$ gave a torque of $19Nm$. Decreasing the base size to $250mm$ increased the torque to $25Nm$ and a base size of $200mm$ gave $27Nm$.

The measurements of the surface sizes in the mesh scene gave an estimate of the cell and surface sizes which was then altered separately. Surface mesh values for the velocity inlet pressure outlet and outer boundary were then chosen as $400mm$ for the relative minimum size and the target size of $600mm$. The mesh values for the interfaces were established as $20mm$ for the relative minimum

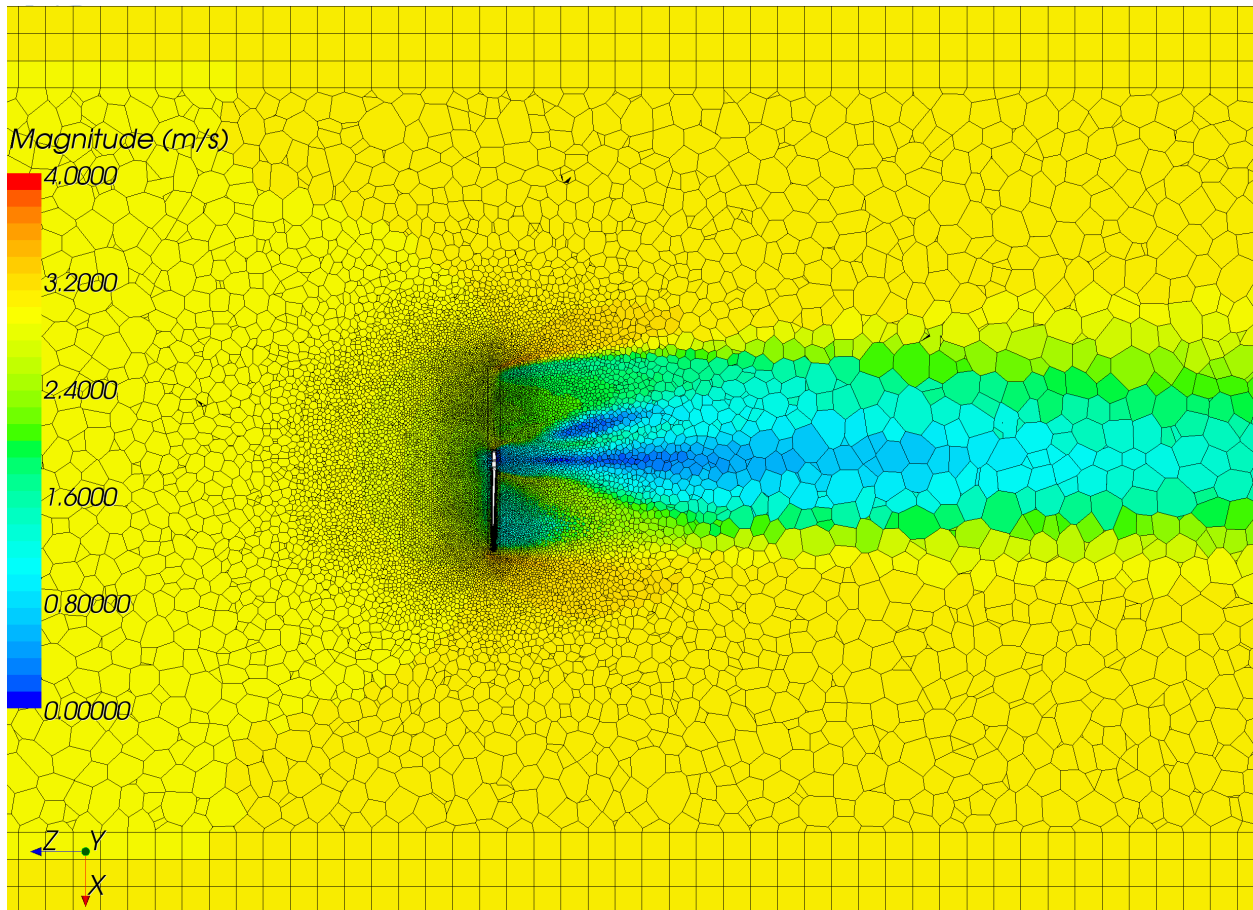


Figure 4.8: Plane section through the center of the domain showing the velocity(magnitude) and volume mesh

size and $30mm$ of the target size. For the blades the mesh size was chosen as $3mm$ for the relative minimum size and the target size of $10mm$. Two boundary layers were chosen with a total height of $2mm$. A volumetric cylinder was inserted for the wake region at the back of the wind turbine. This size was chosen as $20mm$. This increased the torque significantly to $42Nm$. Increasing the boundary layer to $10mm$ with 2 layers decreased the torque to $40Nm$. This indicate that the boundary layer should have more layers. The torque curve against iterations for the previous and the alternations to follow is set out in Figure A.5 (this proses was between 0 and 6100 iterations).

The trailing edge has a small thickness, therefore the trailing edge's cell size should be specified independently from the blade. As flow over the trailing edge is negligible the prism-layers was disabled and the mesh size was chosen as $0.2mm$ for the relative minimum size and the target size of $1mm$. The cell size on the rest of the blade was decreased to $0.1mm$ for the relative minimum size and the target size of $8mm$. This increased the torque to $46.1Nm$ with a total of 5.5915 million cells.

The cell size of the blade was decreased to $0.09mm$ for the relative minimum size and the target size to $4mm$. This decreased the torque to $45.5Nm$ with a significant increase in cells to 7.437

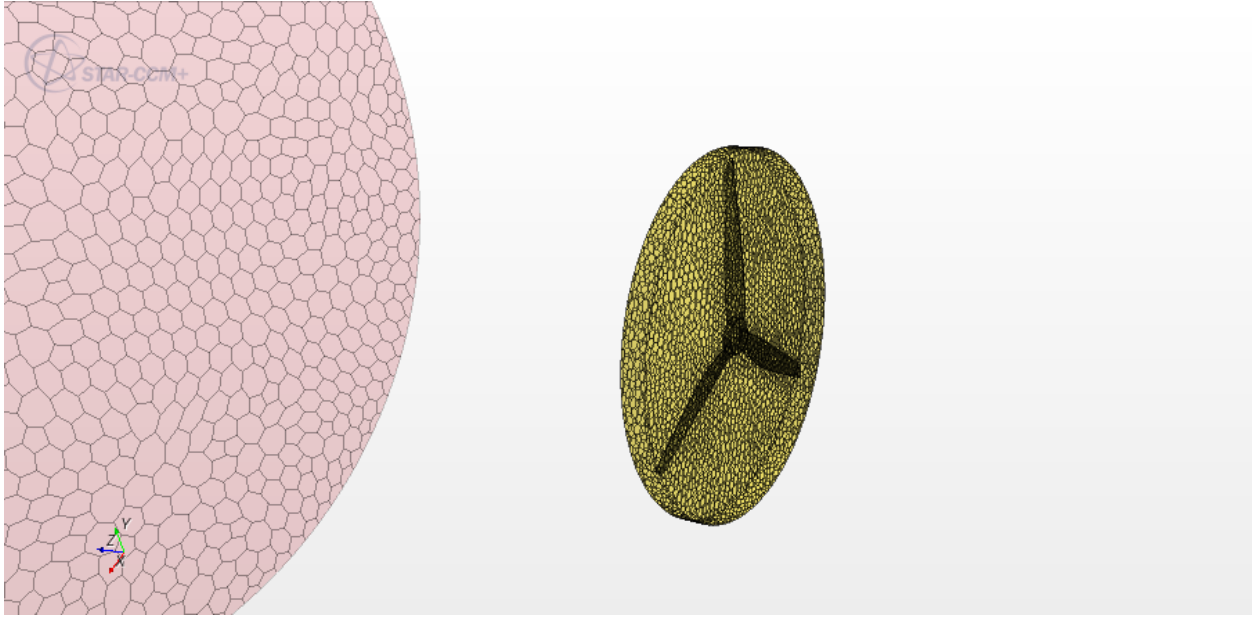


Figure 4.9: Surface mesh of the wind turbine model, interfaces and inlet boundary

million.

Next the trailing edge mesh size was changed to $0.1mm$ for the relative minimum size and the target size to $0.4mm$. The prism-layers on the surface of the blade were increased to 15 with a thickness of $8mm$. The torque increased to $45.75Nm$.

The boundary layer was reduced to $7mm$. This required the minimum thickness of the boundary layer cells on the surface of the blade to be altered to $0.6123mm$, that increase the number of volume cells. To reduce the number of cells the hub area that was discretized with the same volume elements as the blade, was split. The hub's prism-layers were disabled and the cell size was increased to $0.1mm$ for the relative minimum size and the target size of $30mm$. The total number of cells increased to 13.8 million after the volume mesh was generated. This reduced the torque to $43Nm$.

A cone with volume shapes was inserted in the wake region to determine if the region was sufficiently discretized with cells. After the cells in the specified area were measured, a cell size of $100mm$ was chosen. The total cells increased to 14.7 million cells with a decrease of $0.7Nm$ in torque.

As this reduction was small for a increase of 900000 cells, the interface, blade trailing edge and hub area were altered to smaller surface sizes in an attempt to reach cell independence. Interfaces were changed from $0.02m$ for the relative minimum size to $0.015m$ and the target size from $0.03m$ to $0.025m$. The target size of the blade was reduced from $8mm$ to $7mm$. The hub was changed to $0.1mm$ for the relative minimum size and the target size to $20mm$. Trailing edge values were reduced to $0.08mm$ for the relative minimum size and the target size to $0.4mm$. The total number of cells

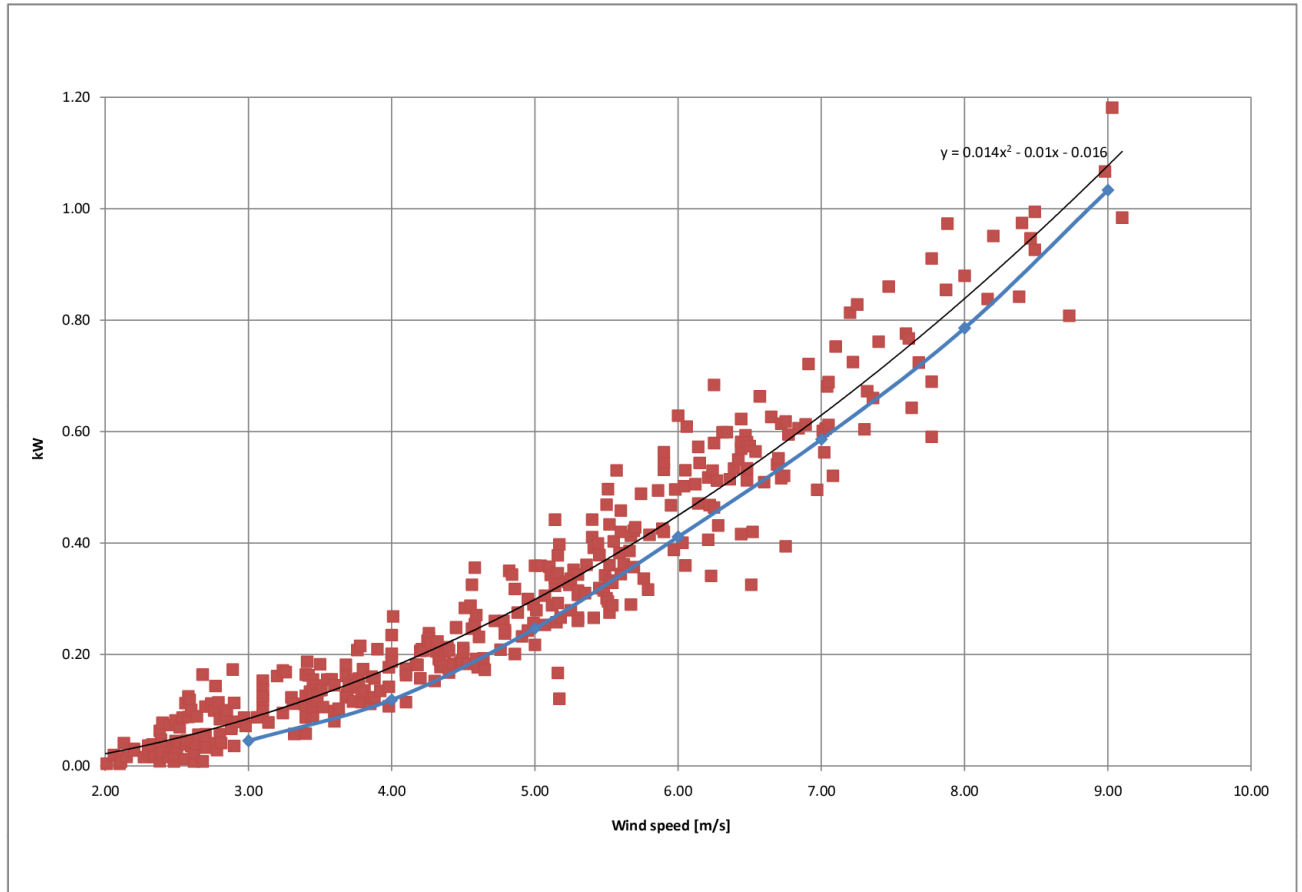


Figure 4.10: Power curve for simulated values and measured values for the open wind turbine

increased from 13.8 million ($42.987Nm$) to 19.4 million ($42.9955Nm$). These results demonstrated that cell independence was sufficiently reached at 13.8 million cells.

4.4.4 Reflection on results and CFD solve information of the wind turbine simulations

The 13.8 million cell simulation was then used to simulate the wind speeds with omega as tabulated in Table 4.1. This, together with the given rotation speeds was used to plot a power to wind-speed curve (Figure 4.10). A generator eff of 88% was incorporated to the simulated results before the plot. The red blocks are the measured values for the wind turbine for differed wind speeds. The blue line is the simulation results for the simulated wind speeds. The simulation results follow the trend of the measured values indicated by the black line.

Figure 4.11 shows the wind velocity and mesh of a section perpendicular with the radial direction of one of the blades. The influence of the near core aspect ratio on the thickness of the boundary layer is evident in the leading edge region. The velocity increase on the upper surface of the blade is also visible. This also supports that the blade was rotating in the correct direction (counter

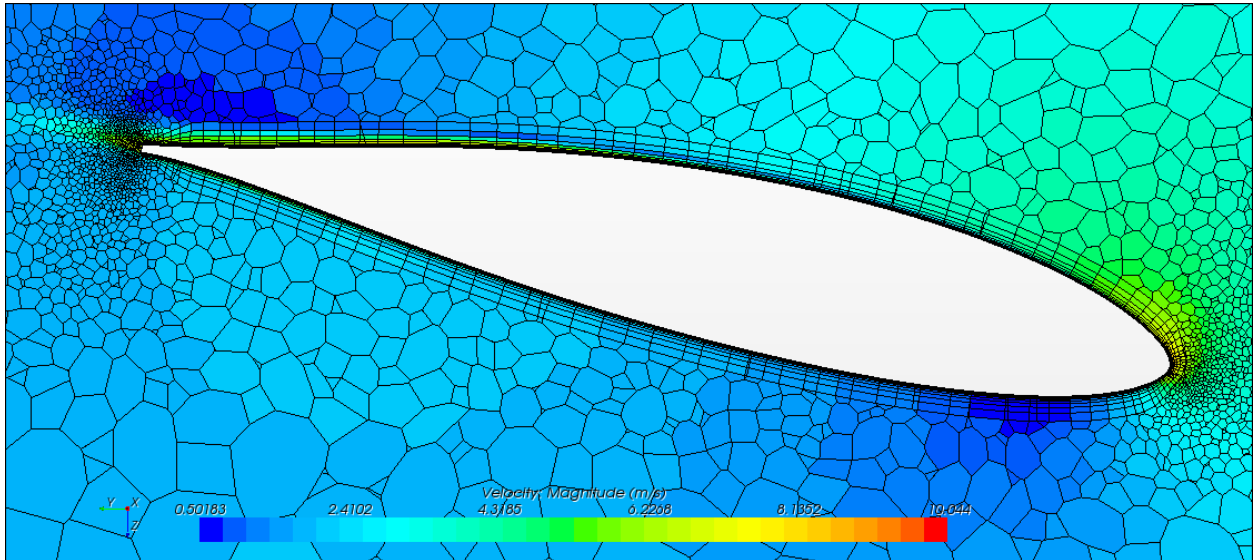


Figure 4.11: Plane section of one blade perpendicular with the radial direction

clockwise) as previously specified in the simulation with the right hand rule.

The velocity (magnitude) at a inlet air speed of 3m/s and the mesh for the simulations are depicted in Figure 4.8.

The Wall Y^+ values are shown in Figure 4.12. The values are smaller than 1, as preferred for the type of flow over the blade.

The momentum monitor plot for the different wind speeds is presented in Figure A.5 (between 6100 and 13900 iterations).

The residuals for the validation process are set out in Figure A.6.

4.4.5 Summary of the wind turbine simulations

The simulated values are lower than the least squares fit, but never lower than any of the measured values. Simulating another wind turbine in exactly the same way, would give conservative values for the power to be produced.

4.5 Summary

For some of the residuals the value was above 0.001. In order to decrease this below 0.001 required a great increase in computing memory and time that was not available. The influence of these higher residuals was tested in a program with high computing cost and the higher value did not influence the velocity in the shroud and the torque on the simulated blades. This problem only occurred when the transition model was introduced.

The simulation results confirmed that the CFD modelling of the shroud and the open wind turbine do sufficiently reflect the reality.

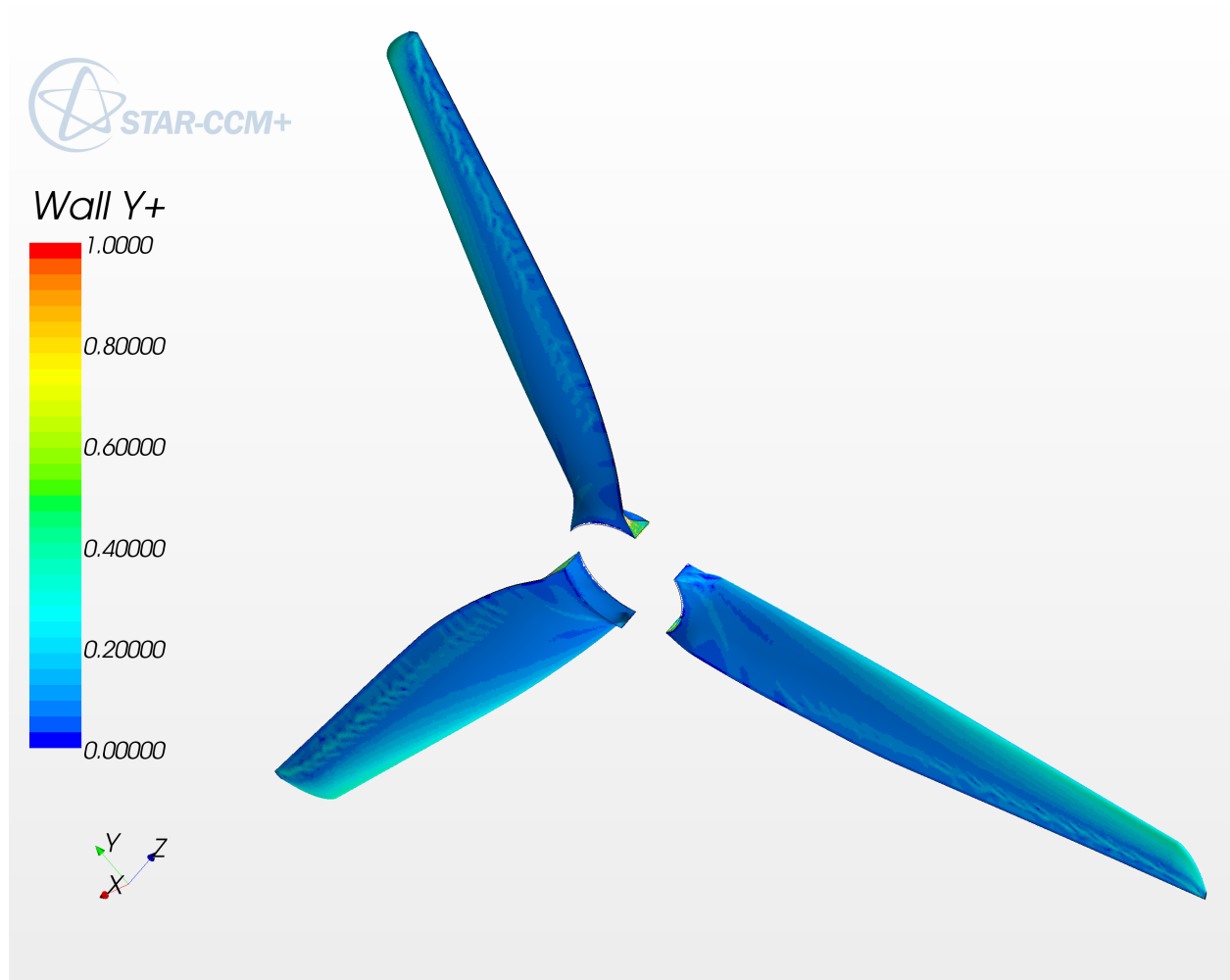


Figure 4.12: Wall Y^+ values on the surface of the blades

Chapter 5

Design

5.1 Introduction

This chapter is divided into two parts, namely the aerodynamic design of the shroud and the turbine which is situated in the shroud. The shape of the brim and the diffuser was not optimised as criteria from Ohya et al. (2008) was used to design the shroud. The diffuser was first designed without a wind turbine inside. After the compact diffuser arrangement was designed measurements of the air velocities was made within the CFD model. From these measurements a wind turbine was designed for the new compact configuration.

5.2 Shroud design

5.2.1 Parameters for the shroud design

The configuration's total diameter, D_T as depicted in Figure 5.1, had not to be larger than $3.6m$ in order to be able to compare it with the open wind turbine *AE 1.0kW* of Bosman et al. (2003). D_T was the reference diameter to compare the measured power generated by the open wind turbine (*AE 1.0kW*) with a shrouded wind turbine with the same total diameter.

From the literature survey in Chapter 2.4, the importance of a diffuser with inlet nozzle and a brim were shown. Ohya et al. (2008) tested a pole mounted wind turbine with diffuser and brim. These ratios were selected for implementation to the new design as it proved well tested. Tests showed that an angle of 4° (Figure 5.1) for the diffuser was the most effective angle. It was noted that a long diffuser had a "remarkable" increase in flow velocity at the inlet of the diffuser but from a practical viewpoint a diffuser with a ratio $\frac{L_1}{D_1} < 2$ was proposed as depicted in Figure 5.1. An inlet shroud with length (L_2) 0.22 times the diffuser inlet diameter (D_1) was used with a total inlet diameter(D_2) of 1.14 times the diffuser diameter(D_1). With the development of the brimmed diffuser the total diameter (D_T) of the configuration was not considered. The optimum brim length

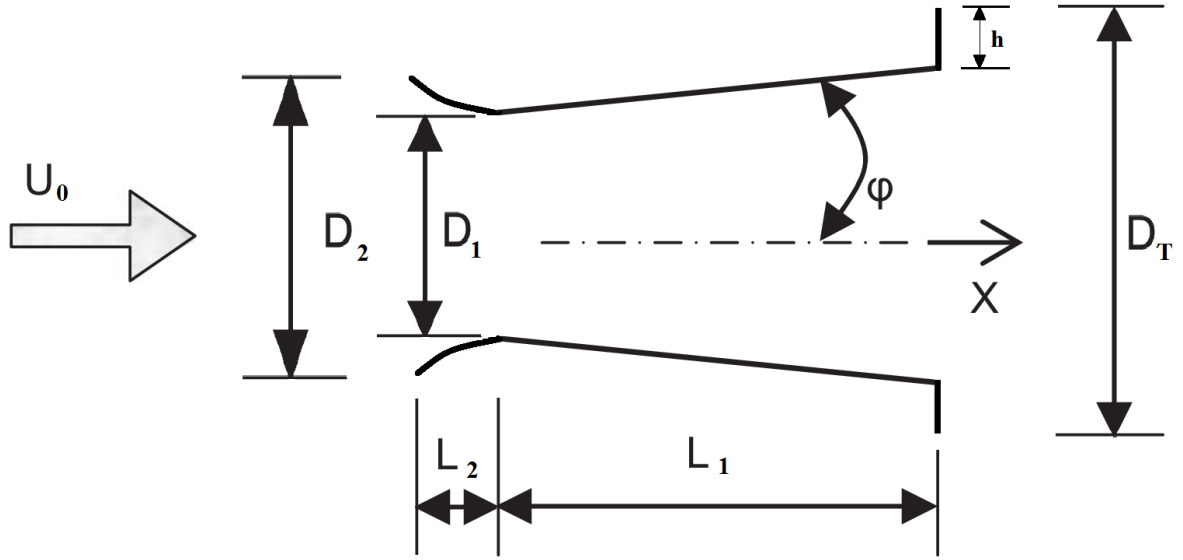


Figure 5.1: Diffuser dimensions

(h) and diffuser angle was determined after an inlet diameter was specified. Therefore $\frac{h}{D} = 0.125$ was chosen although it was not the optimum brim height as seen in Figure 4.2. This would have increased the inlet diameter and thus the area of the wind turbine in the arrangement.

This resulted in a diffuser length of $4.83m$ (L_1 in Figure 5.1) with an inlet diffuser diameter of $2.47m$ (D_1), an inlet nozzle with a length of $0.542m$ (L_2) with a diameter D_2 of $2.815m$ and a brim height of $0.309m$ (h) to obtain a total diameter of $3.6m$ (D_T).

The airflow in the brim was concentrated to the wall of the diffuser with the use of a revolved airfoil body as depicted in Figure 5.3. The use of this body for a center hub reduces drag and thus increased the flow of air through the diffuser. A thick low Re airfoil was chosen for the inner concentrator. An Eppler 862 strut airfoil with a maximum thickness of 32.4 % at 28.5% chord was chosen. The highest point was inserted at the point in the diffuser with the highest velocity at the center as shown in Figure 4.2, at $x/L_1 = 0.225$ from the diffuser inlet. At this point the diffuser's radius is $1.291m$. The scale of this revolved airfoil body was altered in order to increase the diameter of this inner concentrator during the optimization process in CFD as elaborated on in the section 5.2.3.

5.2.2 Shroud simulation set-up

The same set-up as was used in the validation process was applied. A sixth (60°) of the diffuser was simulated with the radial walls modelled as symmetry planes to reduce computing time. Initially the same surface sizes were used as found after cell independence was reached in the simulations of the shroud in Chapter 4.3.3. This volume mesh was used to simulate the increase in diameter

Radius of concentrator	Velocity in annulus	Available power
m	m/s	with $\frac{1}{2}\rho AV^3$ (W)
0.308	3.484	104.5
0.5	3.779	120.1
0.64	3.979	124.5

Table 5.1: Total available power with increase of the radius of the concentrator

of the inner hub concentrator as shown in Figure 5.3. Mesh resolution were then altered until cell independence was reached with a similar process as were done in Chapter 4.3.3. The shroud was then simulated as depicted in Figure 5.4, with more computing cost, as the flow velocity value obtained was used to design the wind turbine blades in the shroud. The domain was divided into an outer, center and inner regions, this was done to introduce mesh resolutions in certain regions to obtain cell independence with less computing power. Wall Y^+ values were also taken into account, as well as residual values as surface mesh sizes were altered. The inlet velocity was chosen as $2m/s$ to model a low wind speed condition.

5.2.3 CFD analyses of the shroud design

Several iterations with varying sizes of the inner concentrator was simulated. A radius of $0.308m$ was assumed to start the first simulation, after which the the radius were altered as seen in Table 5.1. The point of maximum thickness of the concentrator was initially kept at $x/L_1 = 0.225$ from the diffuser inlet as refer to in Section 5.2.1. The average velocity over the annulus between the diffuser and the highest point of the concentrator was then measured and the available powers were determined with Equation 3.21 as shown in Table 5.1.

From Figure 5.2 is is evident that the available power increased significantly as the radius increased from $0.308m$ to $0.5m$. From $0.5m$ to $0.64m$ the rate at which the power increased with changing the radius reduced. With the insertion of blades the annular area would also reduce, therefore the increase of the radius to $0.64m$ was seen as sufficient.

In Figure 5.3 one can see that the two regions of high velocity were not aligned. The highest point of the concentrator was thus moved until the maximum thickness of the airfoil was at the inlet of the diffuser, to align the high velocities. The leading edge then pointed beyond the inlet nozzle and reduced the airflow into the diffuser. Therefore the inlet nozzle was lengthened to align with the leading edge. Figure 5.3 demonstrates the separation of the airflow from the boundary layer at the trailing edge. The trailing edge was then extended to the exit of the diffuser to reduce drag and thus increasing the flow of air into the flow region as depicted in Figure 5.4. From the two figures it is evident that as the trailing edge of the airfoil was extended, the low velocity region

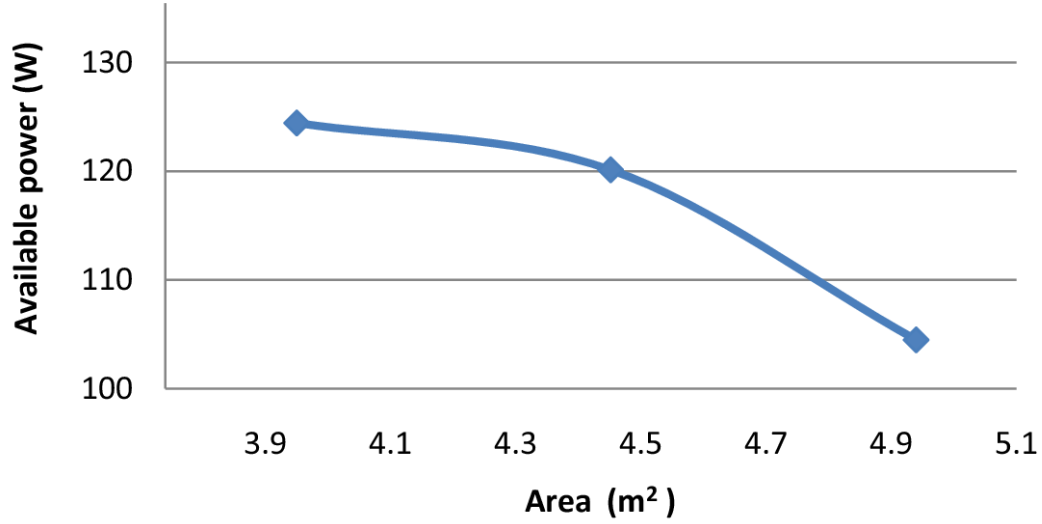


Figure 5.2: Available power as the inner concentrator radius increased with a decrease in flow area inside the diffuser

at the back was reduced. The average velocity was then measured for the annular surface between the maximum thickness of the airfoil and the inlet of the diffuser as depicted in Figure 5.4 and it increased from 3.979m/s to 4.493m/s . The available "kinetic" power increased to 158.23W from 124.45W . As this was almost 4 times the power than the open configuration it was seen as sufficient to test a new shrouded wind turbine configuration.

5.2.4 CFD solve information of the shroud design

The Wall Y^+ values are set out in Figure B.1 and the residuals illustrated in Figure B.2, these values were acceptable. The influence on the residuals after 5000 iterations is evident from the implementation of the boundary layer model in Figure B.3. Tested in simulations the high values for some residuals did not influence the obtained velocity values in the shroud. An increase in the surface average velocity after the 5000 iterations is revealed in Figure B.3. The introduction of the boundary layer model showed a lower drag value and therefore also a higher value of flow into the shroud.

5.2.5 Summary of the shroud design

The available power is 41W at a wind speed of 2m/s ($P = \frac{1}{2}\rho AV^3$) for an open wind turbine and the available power is 158W ($P = \frac{1}{2}\rho AV^3$) for the concentrator diffuser arrangement for the same wind speed. This shows that this configuration is an effective method to accelerate the wind towards a wind turbine.

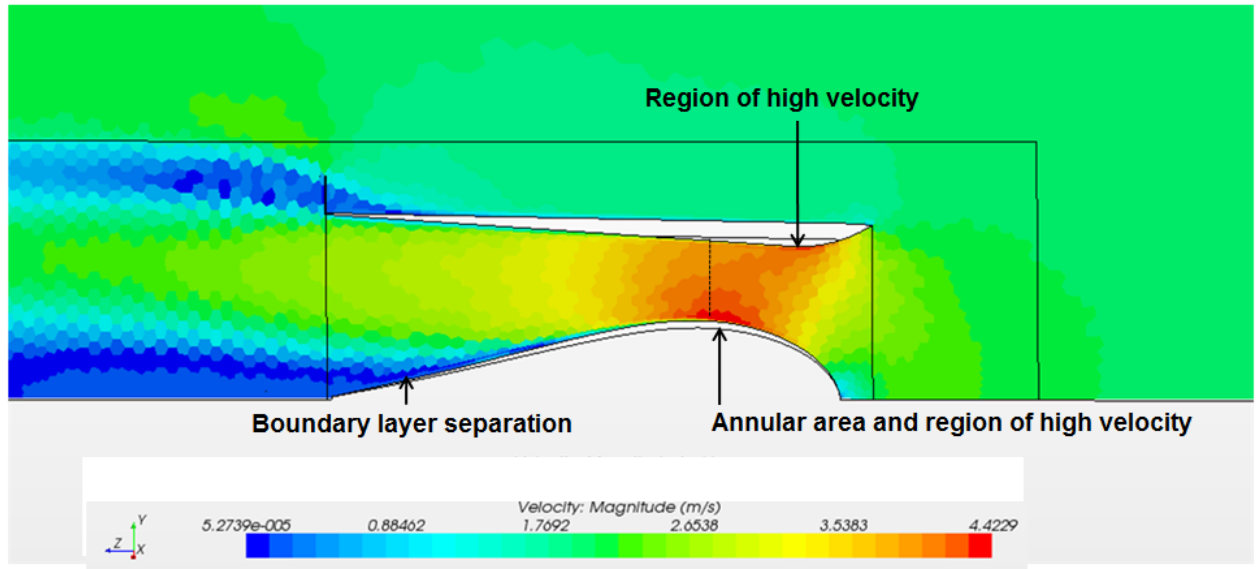


Figure 5.3: Velocity(magnitude) in the shrouded diffuser with brim and revolved airfoil concentrator

5.3 Blade design

5.3.1 Introduction to the blade design

The focus of this study was to determine if a shrouded wind turbine could be beneficial for low wind speed. The focus was not to design a optimum blade. Therefore much of the criteria of the open wind turbine of Bosman et al. (2003) was used to design the blade, as the power to wind speed curve was also used to compare with the shrouded wind turbine.

With a short blade and with tip losses almost nullified, a adapted blade element theory approach was used to design the blade. The blade angle was then chanced to a more optimum angle in the CFD simulations.

5.3.2 Parameters for the turbine blade design

The airfoil that was selected, was the same airfoil as the open wind turbine that was simulated in the validation chapter. This airfoil of the *AE 1kW* from Bosman et al. (2003) was designed for low wind speed regions. This was an obvious choice as the *AE 1kW* open wind turbine is compared with the new shrouded wind turbine in Chapter 6.

A tip speed ratio of 5 was then chosen for the blade design. The local speed ratio and total velocity (U_T) in Figure 3.2) at each blade element was calculated with this tip speed ratio and the average velocity in the shroud in the part where the wind turbine would have been situated. This velocities and the inductions factors determine the blade angle as depicted in Figure 3.2. After the completion of the blade designed, tip speed ratio against power coefficient graphs were drawn from torque's obtained from the CFD modelling. The blade angle was then optimised that once again

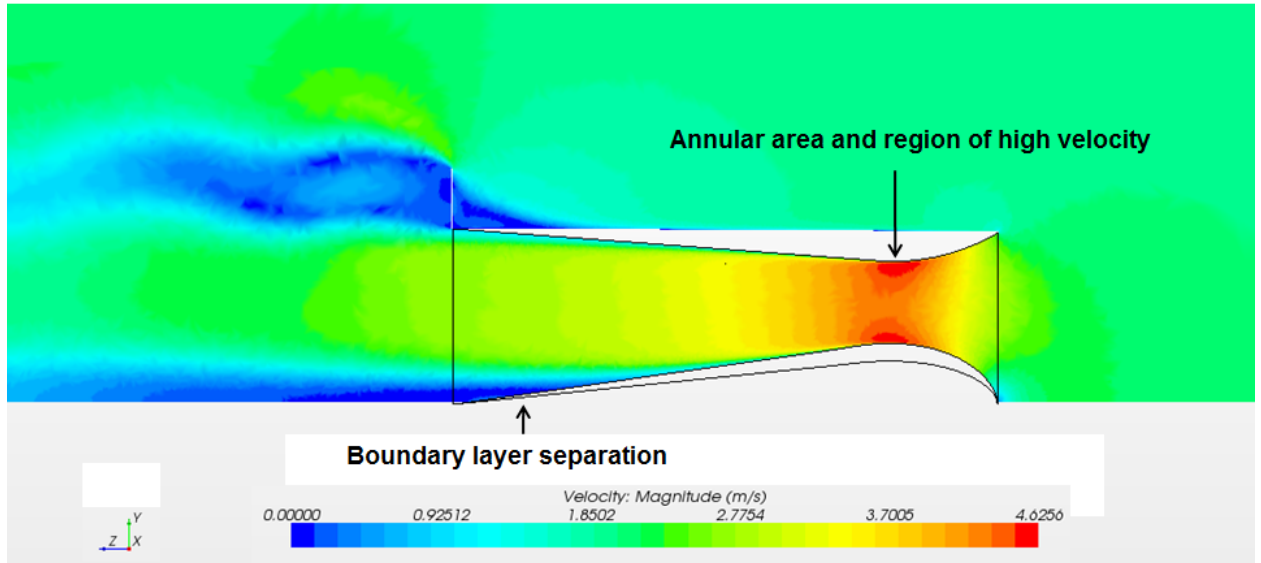


Figure 5.4: Velocity in shroud with airfoil concentrator moved towards the inlet

changed the tip speed to a more optimum level for a highest C_p value, therefore the choice of 5 was from the beginning not perceived to be optimum.

The blade was designed at a free wind speed of 3.5m/s , which is the peak C_p value of the *AE* 1kW wind turbine as seen in Figure B.4 at maximum efficiency. This was done as the results of these two wind turbines is compared in Chapter 6. In the case of other wind speeds, this specific blade design will have an influence on the efficiency therefore the peak power coefficient and tip speed ratio as it is a blade with no pitch control. From Chapter 5.2 the air flow velocity without a turbine in the shroud was 8.07m/s for a simulated wind speed of 3.5m/s . The root radius was 0.64m and the tip radius was 1.233m . The density of the air was set as 1.005kg/m^3 for the elevation of Potchefstroom at 25°C .

A number of three blades were chosen to design the turbine, the same as the *AE* 1kW from Bosman et al. (2003).

The maximum power coefficient was chosen as 0.5 regarding the max coefficient of the *AE* 1kW (Bosman et al. 2003). The blade in the shroud had a very short length, with a cord that starts at $\frac{r}{R} = 0.52$. Figure 2.3 indicates that the induction factors would not have varied much for an ideal wind turbine above this ratio. For a non ideal wind turbine with N number of blades and losses this induction factors will variate more significantly. Therefore at the blade tip a Prandtl's tip loss factor is used to correct the induction factor (Wood 2011, 77). To determine the variation of induction factors an iterative process is followed to determine the torque and thrust on the blades

(Wood 2011, 47), for a number of blade elements. As the wind turbine is situated in a shroud it was not necessary to follow the iterative process as tip losses would have almost been nullified. Also, the extreme short blade involves less variation of induction factors. As determined in section 5.2, a non optimum wind turbine would still have given a high power output as the predicted available power is 4 times higher than a open wind turbine. Given this as well as the fact that it is unknown how much the velocity will reduce in the shroud with the implementation of a rotating turbine, the induction factor for the length of the blade was set as $\frac{1}{3}$. An average angular induction factor (0.019) was determined for the blade in EES and tested in the spreadsheet for its minimum and maximum. The results showed that it had an influence of 0.14° on the blade angles at the root and tip (as the blade is very short). The angular induction factor was then set at 0.019 for every blade element.

Within the CFD simulations the optimum C_p value were found by changing the value of θ_p , that ultimately changed the sum of the two inductions factors components as depicted in Figure 3.2. Therefore it was not felt necessary to have a variation of the induction factors.

5.3.3 Blade element theory design

For a blade design the blade is divided into a number of blade elements. Wood (2011, 41) proposed 10 to 20 blade elements for the blade of a small wind turbine. For this study's large hub region and short blades, 10 elements were selected to design the blade.

Figure B.7 illustrates the geometry of the blade as determined through the above values. Column 16, the max torque per blade element was determined with Equation (3.25). The annulus in the shroud was divided into 10 annular tubes and each of these were able to deliver a maximum torque. The angle column 13 was determined with Equation (3.26). This together with the chord length determined the torque (column 15). Every time the cord length was changed, the Re changed. The spreadsheet was frequently updated with the changes. This allowed for the Re and the two dimensional lift and drag coefficient as well as the angle α (Figure 3.2) of the blade to be determined. The process was repeated with a change in chord length until column 15 resembled the value of column 16. This was repeated for every annulus area (each row). It is clear that the total for the two columns is a close match. Equation (3.27) was used to determine column 15.

The Re and airfoil was applied to determine the two dimensional drag and lift coefficient with XFLR5 (Figure B.5 and Figure B.6). This was continuously updated in the iterative process in the previous paragraph for every annular area. The C_l used in column 11 (Figure B.7) was read from the graph for the peak C_l/C_d values of the specific Re. One can see that the maximum lift to drag ratio is at $\alpha = 7^\circ$. The chord length was then changed in this proses until column 15 resembled the value of column 16. The blade angle ϕ (column 14) together with the chord length were used to draw the blade. Figure 5.5 shows the cord and Figure 5.6 illustrates the twist, with θ_p as seen

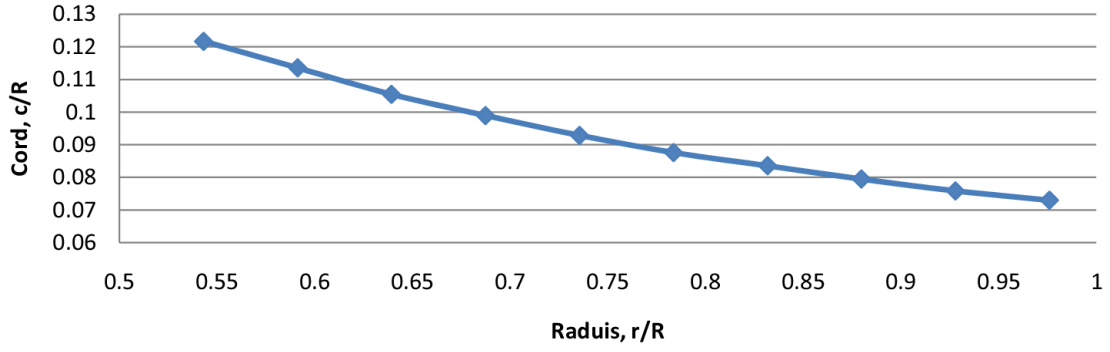


Figure 5.5: Cord of the designed blade

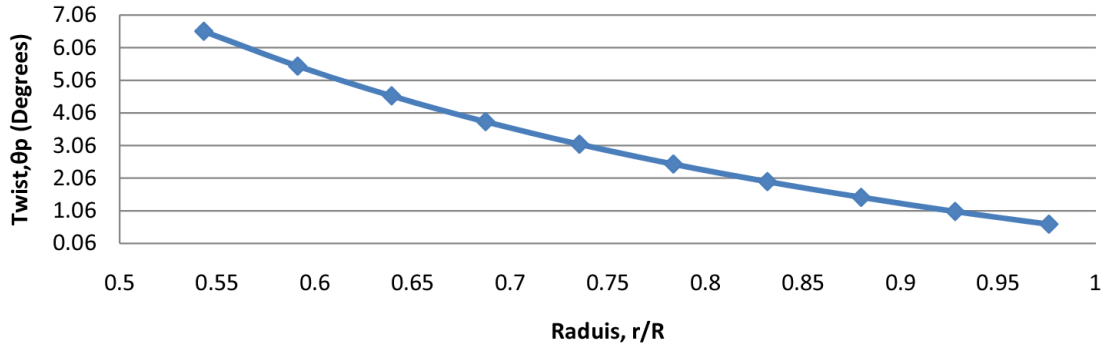


Figure 5.6: Twist of the designed blade

in Figure 3.2 at a $\alpha = 7^\circ$.

5.3.4 Wind turbine simulation set-up

The same domain were used as simulated at the end of Chapter 5.2.2 as the shroud was already simulated with cell independence reached. The turbine was then introduced into the shroud and then simulated. The mesh values of the simulation of Chapter 4.4.3 was used to create a mesh for the turbine in the shroud. The rest of the set-up is the same as described in Chapter 3.5.

At a tip speed ratio of 7, tip radius $1.233m$ and a wind speed of $9m/s$ the tip speed is $56m/s$, well below the 30% of the speed of sound. The CFD simulations was therefore appropriate done for incompressible flow.

5.3.5 CFD analyses of the wind turbine

The blade that was designed was introduced into the shroud and simulated at the designed rotational speed of $32.7rad/s$. In Table 5.2 the first row indicates the values obtained for the design rotational speed. As the C_p value was very low, the rotational speed (ω) was reduced in increment of $5rad/s$. With a smaller increment value the number of simulations would have increased sig-

Velocity (m/s)	Ω (rad/s)	Torque (Nm)	Power (W)	Tip speed ratio	Power Coefficient
2.86	32.7	0.33	10.8	5	0.0492
3.33	25	1.44	36	3.81	0.164
3.64	20	2.22	44.4	3.06	0.202
4.76	15	1.95	29.25	2.29	0.133

Table 5.2: Simulation results for the designed blade angle

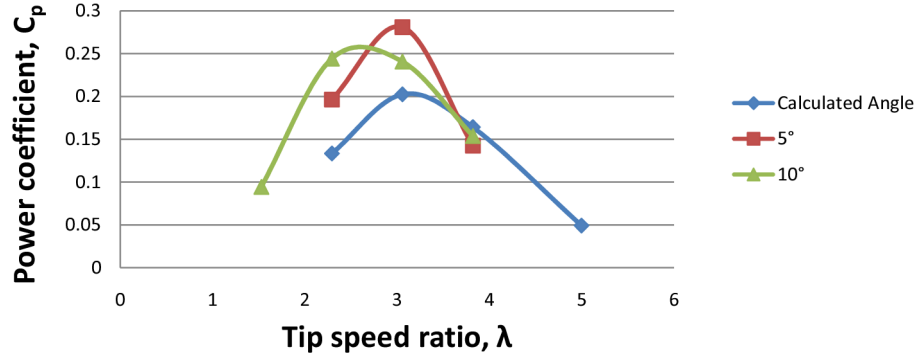


Figure 5.7: Variation of blade angle at a 3.5m/s free wind speed

nificantly. This choice would have produced non optimum values, but is sufficient to show if the configuration has potential, considering that the power produced should at least be 4 times higher than the open configuration. The highest power output was then reached at an omega of 20 rad/s with a C_p of 0.202 as calculated with D_T as shown in Figure 5.1. The C_p values were at a generator efficiency of 100%.

The low $C_{p,Max}$ value and a small blade angle θ_p , lead to an alteration of the blade angle with increments of 5°, also chosen as previously elaborated upon in the previous paragraph. The highest $C_{p,Max}$ value for this showed an increase of 5° (represented in Figure 5.7 which caused a $C_{p,Max}$ value of 0.281 with a tip speed ratio (air speed in shroud, without wind turbine) of 3.06 as revealed in Table 5.3. This was substantially higher than the C_p of 0.202.

In Table 5.2 and 5.3 the first column presents the air flow velocity that was measured in the simulations at the front of the blades during its rotation. Without the turbine the air velocity was 8.07m/s for a simulated wind speed of 3.5m/s.

The C_p value of 0.281 is lower as the C_p value of 0.48 for the *AE 1.0kW* wind turbine of Bosman et al. (2003) as seen in Figure B.4. As it was felt that al the available energy was not extracted, the number of blades was increased. For the 5° increase of θ in the design blade angle a simulation was done with 6 blades for the same wind speed of 3.5m/s. This resulted in a maximum power of 48.6W and a $C_{p,Max}$ value of 0.221, lower than the three bladed simulation with the same blade that gave a C_p of 0.281. The higher solidity caused these values to be lower than the configuration

Velocity (m/s)	Ω (rad/s)	Torque (Nm)	Power (W)	Tip speed ratio	Power Coefficient
4.8	25	1.25	31.25	3.82	0.143
5.1	20	3.08	61.6	3.06	0.281
5.6	15	2.87	43.05	2.29	0.196

Table 5.3: Simulation results for an increase of θ_p of 5° on the blade @ a free wind speed of 3.5 m/s with three blades.

5.3.6 CFD solve information for the wind turbine design

Figure B.9 presents the monitor plot and Figure B.10 the residual plot for the design wind speed of 3.5m/s. The number of finite volume cells that discretize the domain was altered to lower the residual values and proved that the measured torque was not influenced. This however influenced the simulation time and therefore the model that produced these plots (Figure B.9 and Figure B.10) was rather applied to determine the values as seen in Table 5.3 for the wind speed of 3.5m/s. This model was used to determine the C_pMax value for the other wind speeds of 2m/s, 5m/s, 7m/s and 9m/s, with the results (generated power) in Chapter 6. The Wall Y+ values is set out in Figure B.8 and is as required.

5.3.7 Summary on blade design

The full simulation results is presented in Chapter 6. The same approach as in Table 5.3 was used for wind speeds of 2m/s, 3.5m/s, 5m/s, 7m/s and 9m/s to determine the power output for the highest C_p value (as depicted in Table B.1 to Table B.5). This was done for the design blade angle θ_p that was increased with 5° , as this angle gave the highest output at the design wind speed of 3.5m/s. A velocity (magnitude) plot for a plane section through the centre of the domain is represented in Figure 5.8. In this representation for a wind speed of 3.5m/s it is evident that the blades is at a velocity of 5m/s. The blades in the shroud is depicted in Figure 5.9.

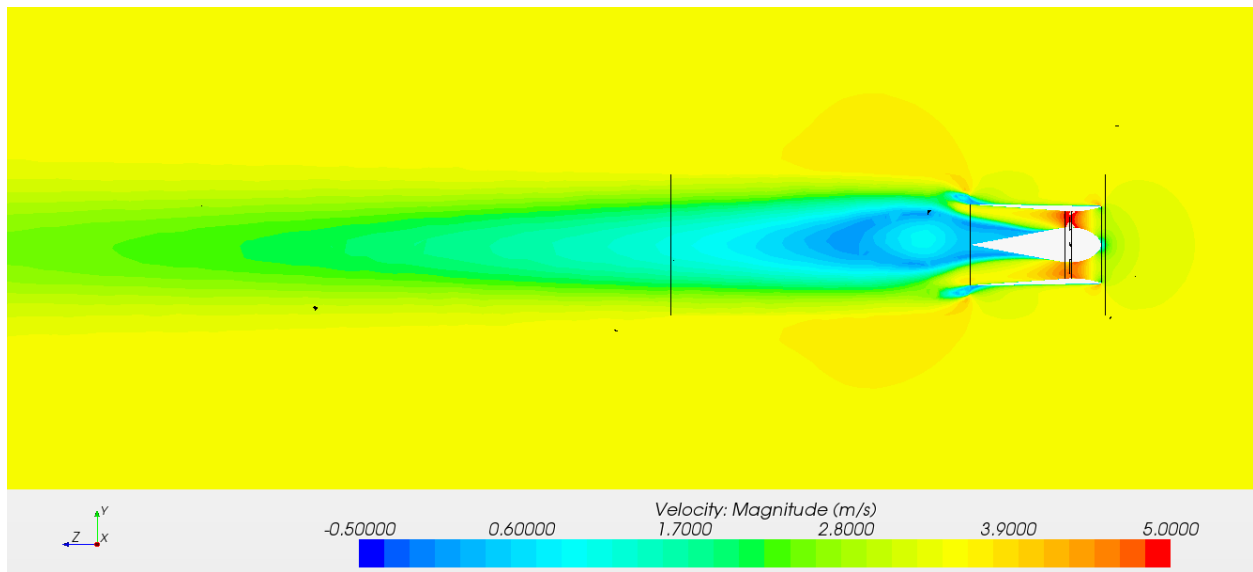


Figure 5.8: Velocity plot of shrouded wind turbine @ 3.5m/s

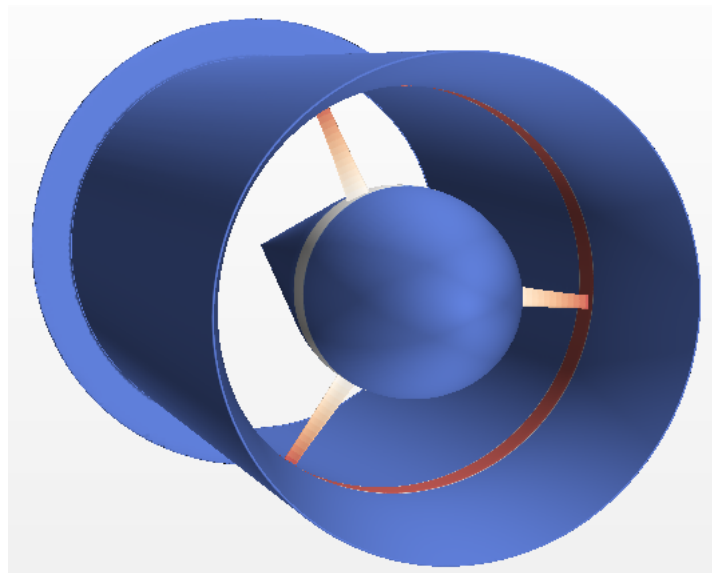


Figure 5.9: Blades in shroud

Chapter 6

Results and supportive theory

6.1 Introduction

The modelling of the new wind turbine configuration presented unexpected results. Therefore, it was necessary to determine a supportive theory for the verification of the results. As this was only investigated after the simulation results were available, the discussion of the supportive theory is included in this chapter.

There was also a reflection on results obtained from the simulations of the shrouded wind turbine and additional simulations at the end of this chapter to further validate the results obtained.

6.2 Results

The results for the CFD simulation was compared with the test results of the open wind turbine of Bosman et al. (2003). The results in Figure 6.1 clearly confirm that the shrouded wind turbine performance was much lower than anticipated and lower than the simulated open wind turbine. At a low speed wind speed of $3.5m/s$, used to design the blade of the shrouded wind turbine, the power produced was $54.2W$ and for the open wind turbine the simulated result was $103.8W$. In the area above $8m/s$ the shrouded wind turbine, power increased above the open wind turbine. This could be attributed to the blades and generator illustrated in Figure C.1. If the power was calculated at a maximum efficiency as in the figure, the open wind turbine would also have outperformed the shrouded wind turbine in this area.

The simulated values included the efficiency of a generator of 88% to compare the results with the measured values. As the power was anticipated to be at least three times higher (shroud design) for the shrouded wind turbine, the basic theory required was deeper investigated which is presented in the next section in order to support the acquired results.

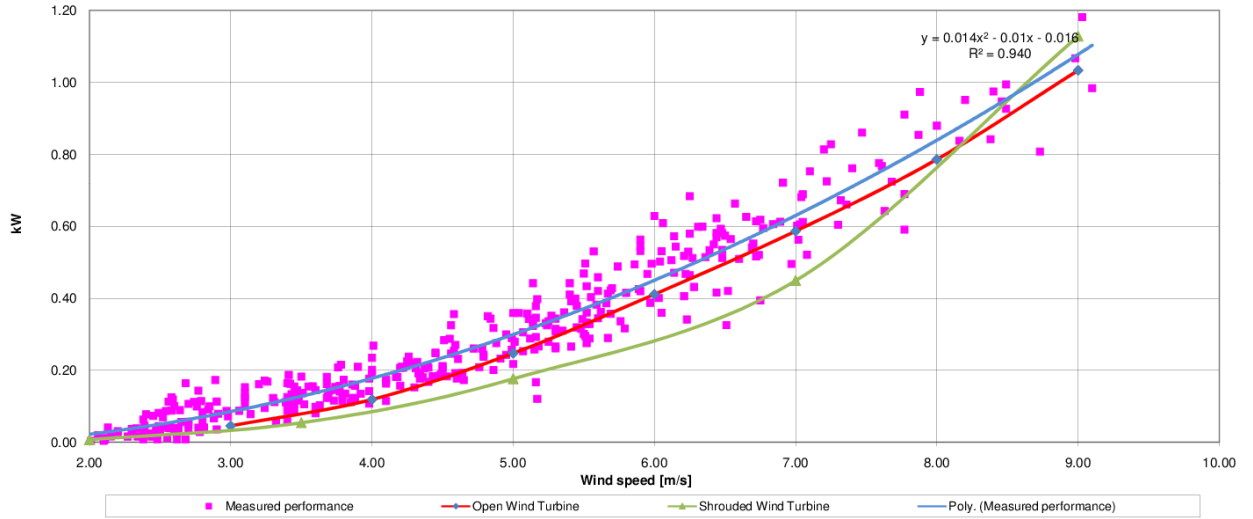


Figure 6.1: Results for the new wind turbine configurations and test results for the *AE 1.0kW* wind turbine of Bosman et al. (2003).

6.3 Available energy and mass flow

After the results showed that the expected performance was much lower than anticipated, fundamental theory was investigated further. A discussion on this investigation and available energy follows.

Figure 6.2 represents a section through a control volume. U_0 is the free stream wind velocity entering the control volume, U_∞ the outlet velocity of the streamtube, R_0 the inlet radius of the streamtube, R_∞ is the outlet radius of the streamtube. The streamlines that flow over the maximum radius of the shroud could be pictured as the boundary of a stream tube (Princeton n.d.). This streamline enters at a radius shorter than the maximum radius and increases in radius as the flow approach the shroud as illustrated in Figure 6.3. The shape of the stream tube is currently irrelevant as the body that it approached will have an influence on it. However, of importance is the fact that the inlet is smaller than the outlet. This has the implication that the velocity decreases at the outlet when energy is extracted (W) from the stream tube. The more the outlet velocity decreases the more energy would be available for extraction. This process is depicted in equation (6.1). This equation is a simplified first law of thermodynamics equation with the following assumptions, namely a steady state, incompressible, isothermal, one dimensional flow with no elevation difference. The static pressure at the boundaries of the control volume was equal to the atmospheric pressure and chosen as $101.3kPa$. The equation and the figure confirm that when the inlet radius is taken the same as the radius at the shroud and the velocity decreases to zero at the outlet of the stream tube, it will give the maximum amount of energy that can be extracted from the stream tube.

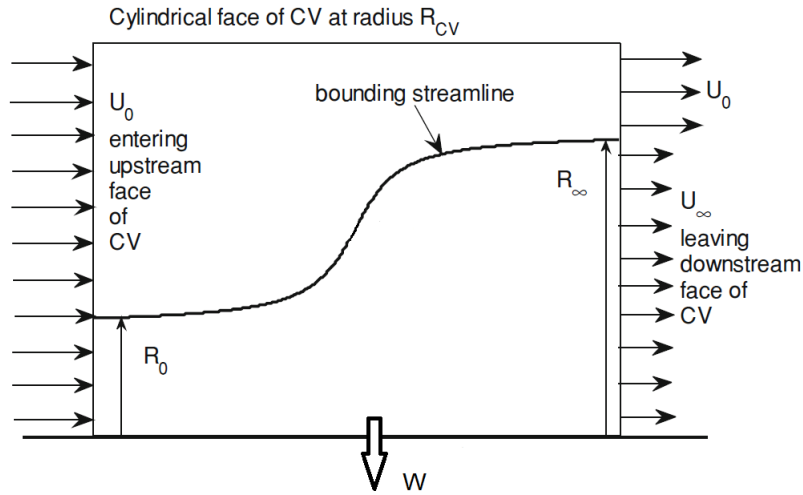


Figure 6.2: Control volume with static pressure and velocities

$$\left(\frac{U_0^2}{2} + P_0/\rho\right) \cdot (\rho \cdot U_0 \cdot A_0) = W + \left(\frac{U_\infty^2}{2} + P_0/\rho\right) \cdot (\rho \cdot U_\infty \cdot A_\infty) \quad (6.1)$$

If A_0 is determined with the maximum radius of the shrouded ($1.8m$) and U_0 is chosen as $2m/s$, the same as the wind speed that was used to design the shroud. The inlet and outlet pressure as atmospheric in (6.1) and the velocity reduce to zero as the outlet radius increases with continuity in mind. The maximum power (W) that could have been extracted from the stream tube reached a maximum of $48.04W$. This value was the same as the described value in Chapter 3 as the maximum power available was determined with $P = \frac{1}{2}\rho A_0 U_0^3$.

Although some researchers thought that an increase in the wind speed in a shroud will increase the energy output above a $C_p = 1$, results confirmed that it is impossible to extract more energy than what is available in the stream tube. The design of the diffuser concentrator arrangement was formulated with a free stream velocity of $U_0 = 2m/s$ and a total outer radius of $1.8m$. This implied a maximum "kinetic power" to be available after accelerating the wind as $158.23W$. The power was determined with $P = \frac{1}{2}\rho A_s U_s^3$, with U_s the air velocity in the shroud and A_s the surface area where the wind turbine should be situated. With the use of (6.1) the power available was $48.04W$, therefore one cannot determine the power available in the shroud with $P = \frac{1}{2}\rho A_s U_s^3$.

$$p_{tot} = p_0 + \frac{1}{2} \cdot \rho \cdot U_0^2 = p_1 + \frac{1}{2} \cdot \rho \cdot U_1^2 = p_2 + \frac{1}{2} \cdot \rho \cdot U_2^2 = p_0 + \frac{1}{2} \cdot \rho \cdot U_\infty^2 \quad (6.2)$$

However, the total power available in the shroud can be determined by considering the total or stagnation pressure. Recall Figure 6.4, which showed the pressure and velocity relation in the free stream of a diffuser. Bussel (2007) concluded that when the flow is present in the free stream, the velocity and static pressure will vary as in Figure 6.4. Also, the total pressure for the inlet, in the diffuser and outlet will be equal if no energy is extracted by a turbine or friction. This was

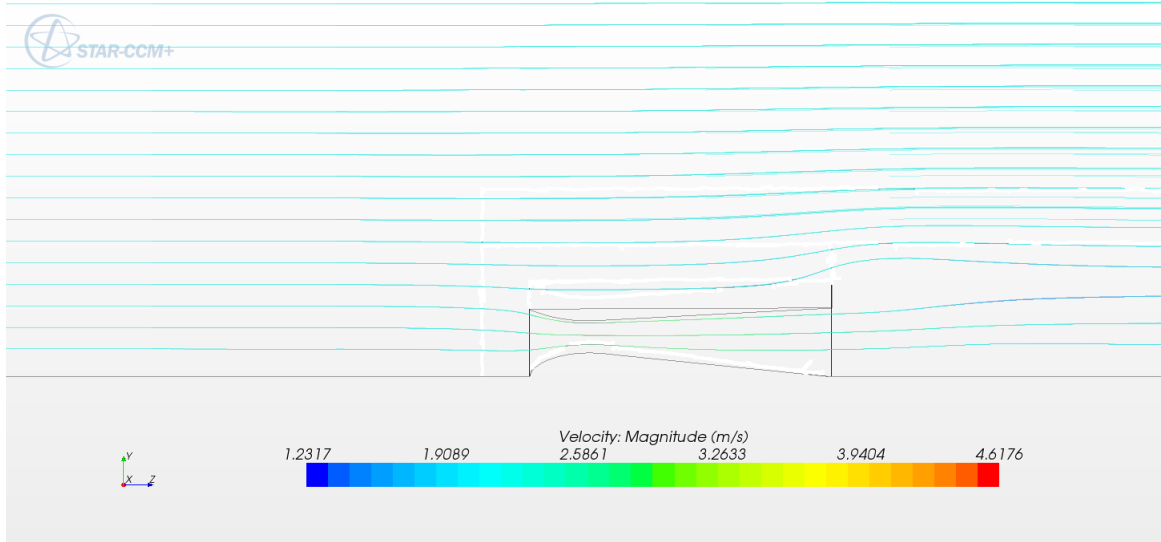


Figure 6.3: Streamlines for the diffuser, concentrator configuration

also measured in the CFD program. A slight decrease in total pressure in the shroud was found as the viscosity effect was included. This would have been the same for the concentrator and diffuser arrangement where the velocity was increased in the shroud with a reduction in static pressure. This is represented with Equation (6.2). If the total pressure is divided with the density one has $(\frac{U_0^2}{2} + \frac{p_0}{\rho})$ to be used in equation (6.1). The adapted Equation (6.3) with variables in Figure 6.4 can be used to determine the available power in the shroud.

$$\left(\frac{U_0^2}{2} + p_0/\rho\right) \cdot \dot{m}_{Shroud} = W_{Shroud} + \left(\frac{U_\infty^2}{2} + p_0/\rho\right) \cdot (\rho \cdot U_\infty \cdot A_\infty) \quad (6.3)$$

If the total pressure at the inlet is the same as the total pressure in the shroud the mass flow in the shroud can be multiplied with the first term $(\frac{U_0^2}{2} + \frac{p_0}{\rho})$. With the velocity at the exit approaching zero and the exit area increasing, the available power could be determined as formerly. For a free stream velocity of $2m/s$ the velocity in the shroud was measured as $4.49m/s$. If the mass flow in the shroud was determined with this velocity and use in equation (6.3) the total power available was $36.98W$, less than the open turbine of $48.04W$. This proves that the mass flow in the shroud is the determining factor to increase the available energy.

If the mass flow for the maximum radius of $1.8m$ with a free stream velocity of $U_0 = 2m/s$ is calculated, the mass flow is $24.02kg/s$. Considering the mass flow for the diffuser concentrator arrangement with the tip radius of the annulus of $1.233m$ and a root radius of $0.64m$ with a velocity of $4.49m/s$ at the annular area the mass flow it is $18.5kg/s$. This shows that some of the mass flow tends to avoid this configuration and have less power available.

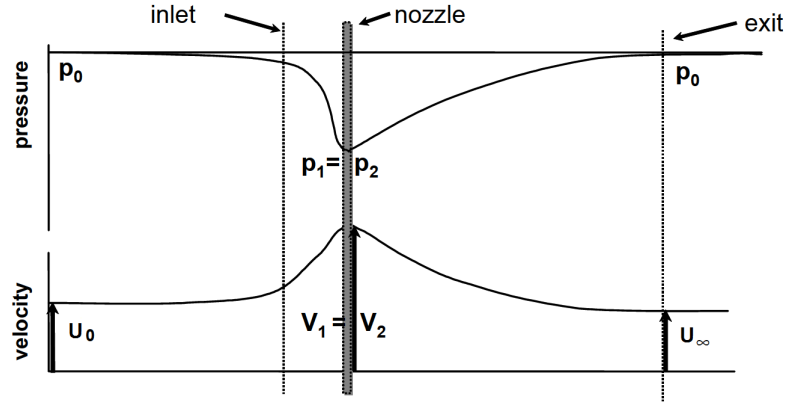


Figure 6.4: Pressure and velocity relations in an empty diffuser Bussel (2007)

Wind speed(m/s)	Air speed in shroud(m/s)	Available power(Watt)	C_p Values
2	4.49	32	0.24
3.5	8.07	174	0.35
5	11.34	497	0.39
7	15.03	1292	0.4
9	20.16	2864	0.45

Table 6.1: New C_p values determined with Equation (6.3) as maximum available power

6.4 Calculated available power and new C_p values

The available power is shown in Table 6.1 for the winds speeds as set out in the power to wind speed curve in Figure 6.1, calculated with Equation (6.3). Also in the table is the values obtained in the simulation of the shrouded wind turbine (plotted in Figure 6.1) and the new C_p values for these values (at a generator efficiency of 100%).

The C_p values increased significantly from a wind speed of $2m/s$ to $9m/s$. This can be attributed to the design and therefore a to long chord length at the lower wind speeds. The blade was designed at $3.5m/s$ with a expectation of available power of $922W$. As the available power is only $173.4 W$ at $3.5m/s$, this would result in a insufficient design and lower C_p value. At a higher wind speed the chord with a longer length would have been more beneficial as the energy available is more. From this then a high C_p value of 0.45 at $9m/s$. This value could have been even higher if the angle ϕ , as depicted in Figure 3.2, was optimum for the wind speed of $9m/s$. A C_p value of 0.45 correspond well to recent develop wind turbines.

Wind speed(m/s)	Available power(Watt)	Available power(Watt)
	Wind turbine with same diameter	Shrouded wind turbine
2	19	32
3.5	103	174
5	300	497
7	824	1292
9	1751	2864

Table 6.2: Total available power for a shrouded and open wind turbine

6.5 Available power for the shrouded wind turbine and a wind turbine with the same turbine diameter

It is necessary to have a perspective for C_p values above the Betz's limit for a shrouded wind turbine. Therefore the total available power for the designed shrouded wind turbine is tabulated in 6.2 with the total available power of a wind turbine that has the same diameter turbine (not the reference diameter).

For C_p values for shrouded wind turbines there is a tendency to use the values of column 2 in Table 6.2 to determine the value with Equation (3.22) with the velocity in the shroud and the wind turbine area in the shroud. This will produce C_p values higher than the Betz's limit as depicted in Figure 2.12. The available power is actually the values in column 3 in Table 6.2, and should be used to determine the C_p values as shown in section 6.4.

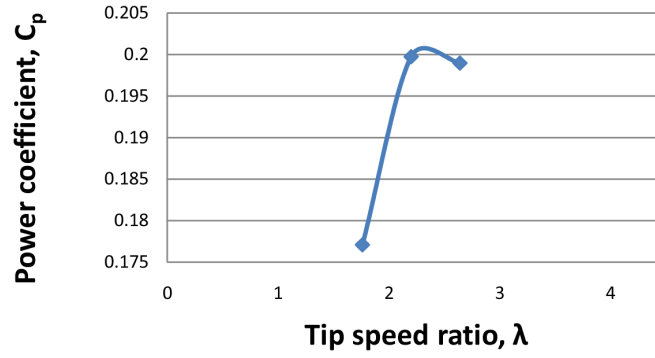
It can be seen for every wind speed the available power is higher for the shrouded wind turbine than for the open wind turbine with the same turbine diameter. Therefore at a low wind speed of $2m/s$ the shrouded wind turbine will have more energy available to start rotating.

6.6 Reflections on results

Next follows a reflection and evaluation of the results that were acquired through the CFD modeling and the values of certain variables that were measured.

For the chosen blade angle that gave the highest $C_{p,Max}$ value, six blades was also simulated although the design was done with three blades. The peak value of this simulation gave $48.6W$ at an $\omega = 15rad/s$. The 3 blade turbine gave $61.6W$ at $\omega = 20rad/s$. The introduction of more blades increased the solidity and reduced the $C_{p,Max}$ value as well as the tip speed ratio. This was expected and described as investigated in the available literature.

For the calculated blade angle in B.7 the average velocity at the front of the blades at the peak C_p value measured $3.64m/s$. For the increase of 5° in θ_p the velocity increased to $5.1m/s$ that is near

Figure 6.5: C_p versus tip speed for the scaled wind turbine

the the induction factor optimum value of $5.38m/s$ ($\frac{2}{3} \cdot 7.08m/s$) for maximum energy extraction. Average velocity of $7.08m/s$ was measured in the shroud before the turbine was introduced. An increase of 10° in θ_p gave an average velocity of $6.1m/s$ at the front of the blades. The velocity near the induction factor velocity produced the highest energy which was expected.

After the blade angle was varied with 5° to determine the angle of highest $C_{p,Max}$ with a free wind speed of $3.5m/s$, the peak C_p value was much lower than expected as the results confirm. This raised a concern that the blade in the diffuser was not correctly designed. Therefore, rather than applying other wind speeds to draw a power curve, the wind turbine of Bosman et al. (2003) was tested in the diffuser. This would have indicated that the simulated power was in range. The wind turbine was reduced in scale to be inserted in the shroud. The scaling would have had a negative influence on the efficiency, but it would have given an indication of the power available. As the available power was perceived to be more than the open configuration and the cord length was less with the change in scale, more blades were introduced. One sixth of the domain was simulated with periodic interfaces to reduce simulation time, with one blade that should effectively give six blades. This was tested with the open three bladed wind turbine that presented results that was 5% lower than the total three bladed simulation. The peak power for this turbine was $43.8W$. At a free wind speed of $3.5m/s$ and a ω of $10rad/s$, it is slightly lower than the concentrator configuration at $48.6W$. This value was 10% lower which indicated that the power available was less than expected and that the simulated power of the concentrator configuration was acceptable. The power coefficient over tip speed ratio (tip speed ratio for the velocity in the shroud) is set out in Figure 6.5. The power coefficient was lower than the coefficients in Figure 5.7. This C_p value should have been lower as Wang & Chen (2008) in Figure 2.6 also have a reduced power coefficient with higher solidity, when more blades were introduced (Figure 2.6).

Testing the significance of the torque, the ω was increased from $10Rad/s$ to $60rad/s$. A diffuser only would have lead to an increase of the air speed to 1.6 times the free wind speed. With

this and the induction factor an air speed of $U_s = 3.5m/s \cdot 1.6 \cdot \frac{2}{3}$ in the diffuser with turbine could have been calculated. A tip speed ratio of 4.4 (tip speed ratio for the velocity in the shroud), a tip angle of 4° for the *AE 1kW* (Bosman et al. 2003) and a radius of $1.233m$ at an omega above $54rad/s$ the blade would have turned into a fan. An omega of $60rad/s$ reduced the torque from $4.38N/m$ to a negative value of $-1.9N/m$. From this it can only be concluded that the turbine was then acting as a fan and power had to be supplied to the blade in order to accelerate the wind. This outcome was expected.

If the blade area is used as reference at $3.5m/s$, then $C_{p,Max} = 0.72$ and $C_{p,Max} = 0.883$ at $9m/s$ at a generator efficiency of 88%, that was in the reach of the shrouded wind turbines that were referred to in Figure 2.6. When the outer area has been taken as reference, a $C_{p,Max} = 0.25$ which increased to $C_{p,Max} = 0.28$ at a free wind speed of $9m/s$ at a generator efficiency of 88%. The design of the cord as too lengthy at the lower wind speed (more power was expected) the value of $C_{p,Max}$ was lower with the effect being reduced in the higher wind speeds. For the *AE 1kW* the field measured power coefficient reduced in the range from 0.468 to 0.234.

The momentum theory was formulated for shrouded wind turbines (Bussel 2007). In this theory, the induction factor to be used to determine the velocity of the air at the front of the turbine blades for an open and shrouded wind turbine, proved to be exactly the same. Thus for maximum energy extraction it can be concluded that the axial velocity of the air at the front of the blades is $2/3$ times the velocity (without turbine) in the shroud where the turbine would have been situated. For the design wind speed the velocity were reduced from $8.07m/s$ to $5.1m/s$ for the peak C_p value, this value was near the induction value of $5.38m/s$. A similar result was obtained for the other modelled wind speeds.

6.7 Summary

The simulated power for the open wind turbine is higher than the shrouded wind turbine if the total diameter is taken as reference.

If the blade area is taken as reference the total available power for the shrouded wind turbine is higher as seen in Table 6.2.

For a open wind turbine the total power available for a wind speed (U_0) before the introduction of the Bezt's limit can be determined from (equation 3.21):

$$P_w = \frac{1}{2} \cdot \rho \cdot A_0 \cdot U_0^3 \quad (6.4)$$

From equation 6.2 for the outlet velocity of the stream tube reducing to zero the equation for the total power available for a shrouded wind turbine reduces to:

$$P_{Shroud} = \frac{1}{2} \cdot \rho \cdot A_{Shroud} \cdot U_{Shroud} \cdot U_0^2 \quad (6.5)$$

With the area A_{Shroud} and U_{Shroud} at the surface in the shroud where the wind turbine should be situated.

A reflection on results, once again validated the values obtained from the simulations. The results therefore can be seen as trustworthy.

Chapter 7

Conclusions and Recommendations

7.1 Conclusions

As seen in Chapters 5 and 6, the power produced by the chosen configuration was less, with a lower C_p value than the values for the open wind turbine, if the total diameter is taken as reference. The methodology followed by other researchers and in this study led to a design with the expectation of a substantial increase in power. As confirmed in the previous chapter, the methodology to increase the velocity in the shroud should not to be done to the maximum. Nevertheless, an increase in the velocity should be tested with theory as elaborated on in Chapter 6.3 to obtain the available energy.

It can be concluded that if the objective was to design a shrouded wind turbine with a higher power output than a open wind turbine with the reference diameter to be taken as the total diameter of the configuration, then the objective was not met. If the surface at the blades was taken as the reference, the C_p value corresponded well with other shrouded wind turbines ($C_p = 0.72$ to $C_p = 0.883$ was at a generator efficiency of 88%). Calculated with Equation (3.22) with the air speed in the shroud and the turbine area in the shroud (As elaborated on in Chapter 6.3). If the C_p values is calculated with the actual available power as described in Chapter 6.4 the values were $C_p = 0.24$ at $2m/s$ and $C_p = 0.45$ at $9m/s$ for a non optimum design. These values reflect well to modern wind turbines. Future shrouded wind turbines designers should use Equation (6.5) to calculate the available power and the C_p values.

For a low wind speed of $2m/s$ and a total diameter of $3.6m$ ($10.18m^2$), without a wind turbine the mass flow is $24.02kg/s$. The mass flow for the diffuser concentrator arrangement is $18.5kg/s$ with the area as $3.49m^2$, where the wind turbine should be situated in the shroud at a average air velocity of $4.49m/s$ in the shroud for the wind speed of $2m/s$. For a total diameter of $2.47m$, the same as the wind turbine in the shroud, at a free wind speed of $2m/s$ the mass flow is $8.23kg/s$. This will imply (Chapter 6.3) that for the small wind turbine in the shroud there is more energy available than the open turbine with the same rotor diameter. Therefore if these two turbines has

the same rotor inertia and generator the shrouded wind turbine will start rotating before the open one and be more productive. The diameter of the two open wind turbines (3.6m and 2.47m) do not differ significantly therefore the starting wind speed should almost be the same. Therefore the shrouded wind turbine will be more appropriate for low wind speed regions than both open designs, as it will start rotating at a lower wind speed. From this it can only be concluded that the total diameter should not be taken as reference.

From the results it is clear that for a free wind speed of 5m/s the blades rotated at a much higher omega (29rad/s) than the open wind turbine (17.88rad/s). This was the case for other wind speeds as well. This should reduce generator size and cost significantly.

Figure 4.8 clearly indicates the increased velocity at the tip of the blade. This resulted in tip losses and less power production. This loss could have been significantly reduced if the turbine was situated in a shroud.

The smaller diameter wind turbine, with a shorter cord (rotating speed) should have a smaller moment of inertia, that could improve the starting speed. This together with the fact that aerodynamic noise could be reduced and safety could be improved, are also seen as design advantages.

The research in this dissertation highlighted the benefits of a shrouded wind turbine. Solutions are proposed in the next section to increase C_p values, productivity at low wind speeds and more optimal design.

7.2 Recommendations and Future studies

From the previous chapter it is clear that the total or stagnation pressure at the far front, back of a shroud and in the shroud is equal if no energy is extracted. If energy is extracted either by a wind turbine or friction in a shroud, it will reduce the total pressure in the shroud. This, together with the fact that the available power is the product of the mass flow and the total pressure that has been divided by the density, it can be concluded that a configuration with a larger mass flow in a shroud will have more energy available. To increase the mass flow it is recommended that a shroud with brim or an airfoil that forms a shroud be tested. The increase of air speed to a maximum is not of importance, but the overall mass flow at the surface where the wind turbine should be situated has to increase. Therefore the possibility of increasing the mass towards a wind turbine with the use of a shroud should be investigated.

Ohya et al. (2008) results from test on a compact diffuser with small brim proves that a high C_p value can be achieved with a compact configuration (Figure 7.1). They achieved a $C_p = 0.54$ with the total diameter as reference. This can be attributed to the small decrease of the total diameter by the compact configuration that could increase the mass flow and reduce tip losses in the shroud. The best known open wind turbine maximum effectiveness is $C_p = 0.53$ (Wood 2011,



Figure 7.1: Compact diffuser with wind turbine (Ohya et al. 2008)

94) which merits further investigation and research. Structural constraints should be considered in conjunction with the aerodynamic design, to develop a shrouded wind turbine that is feasible to be pole mounted.

As with all wind turbines there is a trade off between C_p and the overall productivity (total power production over time). As described in Wood (2011, 94), a narrow performance curve on a variable speed wind turbine requires a very accurate control system to adjust blade speed in order to maintain an optimum tip speed ratio. This peaky curve will also have an influence on starting torque and thus cut in speed and productivity as explained through the literature review. As the open and shrouded wind turbine has its own "character", it is recommended to manufacture the shrouded wind turbine for the purpose of comparing and testing it against an open wind turbine. This is to be done at the same site to determine which configuration produces the most power over a given period of time (referred to as productivity). This will indicate if a new configuration will be superior or not.

It is confirmed that the C_p value at the root area is lower than the tip region. This can be attributed to the higher solidity, lower local speed ratio and a non optimal induction factor (Figure 2.3). Therefore the effect of the inner concentrator could be beneficial as it will increase the radius of the root area and will limit the earlier stated problems. The increase in the hub circumference also results in the possibility to introduce a larger number of smaller, shorter blades. This, could be beneficial as was set out in Chapter 2.3.4. As the concentrator could decrease the overall mass flow, the diameter of the inner concentrator needs to be investigated in a new shroud approach. Overall efficiency and not only the influence on the mass flow should be considered. It is evident that the increase in local air speed in the shroud with the implementation of the concentrator will

increase the local air speed at the blades. The blades will therefore rotate at a higher revolution per minute. This is an overall advantage as the generator will be smaller with less resistive torque and a reduced prize.

If a new configuration can raise the overall efficiency, it will be necessary to design a new wind turbine that has starting torque as focus. Transient simulations in CFD should be done to simulate starting. CFD is an effective tool to design a blade. The local air velocity can be measured at the front of a blade element while the blade is rotating. Furthermore, the induction factors and therefore the local blade angle of the blade element can be adjusted to improve the design.

Small oscillations for the "value of significance" occurred as cell independence was sufficiently reached or with the introduction of the transition model for the boundary layer. This indicated that it is necessary to simulate the domain as transient as there is, for example a vortex shedding behind the brim as depicted in Figure 5.4. As this dissertation focused on steady state with reduced computing resources and time, these simulations were not performed. It is recommended that the values obtained from the transient modelling in CFD be compared with the steady state values.

The C_p values of $C_p = 0.24$ at $2m/s$ and $C_p = 0.45$ at $9m/s$ for the shrouded wind turbine indicate that there is a theoretical limit the same as the Betz limit (Betz 1926) for the available energy in the shroud. The principles of conservation of mass and momentum should be used to determine this theoretical limit or show if it is the same as the Betz limit.

Thus it can be recommended that further research for a shrouded wind turbine should be implemented. This should include aerodynamic design, structural design and the economical feasibility of such a configuration.

Bibliography

- Abe, K., Nishida, M., Sakurai, A., Ohya, Y., Kihara, H., Wada, E. & Sato, K. (2005), ‘Experimental and numerical investigation of flow fields behind a small wind turbine with a flanged diffuser’, *Journal of wind* **93**, 951–970.
- Abe, O. (2004), ‘An investigation of flow fields around flanged diffusers using cfd’, *Journal of wind engineering and industrial aerodynamics* **92**, 315–330.
- AFDB (2004), Newsletter.
URL: <http://finesse-africa.org>
- Babuska, I. & Oden, J. (2004), ‘Verification and validation in computational engineering and science: basic concepts’, *Computer methods in applied mechanics and engineering* **193**, 4057–4066.
- Bernard Frankovic, B. & Vrsalovic, I. (2001), ‘New high profitable wind turbines’, *Renewable Energy* **24**, 491–499.
- Bertin, J. & Cummings, R. (2009), *Aerodynamics For Engineers*, Pearson Education.
- Betz, A. (1926), ‘Windenergie und ihre ausnutzung durch windmühlen’, *Vanderhoeck and Ruprecht*.
- Bosman, J., Jonker, A. & De Bruin, D. (2003), The aerodynamic and structural design of a 1kw wind turbine blade, Technical report, North-West University.
- Burton, T., Sharpe, D., Jenkins, N. & Bossanyi, E. (2001), *Wind Energy Handbook*, John Wiley & Sons Ltd.
- Bussel, J. (2007), ‘The science of making more torque from wind: Diffuser experiments and theory revisited.’, *Journal of Physics* **75**.
- Chen, L., Ponta, F. & Lago, L. (2011), ‘Perspectives on innovative concepts in wind-power generation’, *Energy for sustainable development* **15**, 398–410.
- Giguère, P. & Selig, M. (1997), ‘Low reynolds number airfoils for small horizontal axis wind turbines’, *Wind Engineering* **21**, 367–380.

- Gilbert, B. & Foreman, K. (1983), ‘Experiments with a diffuser-augmented wind turbine’, *Energy Resources* **105**, 46–53.
- Gilbert, B., Oman, R. & Foreman, K. (1978), ‘Fluid dynamics of diffuser-augmented wind turbines’, *Energy* **2**, 368–374.
- Grant, A., Johnstone, C. & Kelly, N. (2008), ‘Urbanwind energy conversion: The potential of ducted turbines’, *Renewable Energy* **33**, 1157–1163.
- Hau, E. (2006), *Wind turbines. Fundamentals, Technologies, Application*, Springer.
- Igra, O. (1981), ‘Research and development for shrouded wind turbines.’, *Energy Convers* **21**, 13–48.
- Langtry, R. B. (2006), A Correlation-Based Transition Model Using Local Variables for Unstructured Parallelized CFD Codes, PhD thesis, University of Stuttgart.
- Manell, J., McGowan, J. & Rogers, A. (2002), *Wind Energy Explained*, John Wiley & Sons Ltd.
- Matsushima, T., Takagi, S. & Muroyama, S. (2005), ‘Characteristics of a highly efficient propeller type small wind turbine with a diffuser’, *Renewable energy* **xx**, 1–12.
- Moriarty, P. & Hansen, A. (2005), Aerodyn theory manual, Technical report, National Renewable Energy Laboratory.
- Oberkampf, W. & Trucano, T. (2008), ‘Verification and validation benchmarks’, *Nuclear Engineering and Design* **238**, 716–743.
- Ohya, Y. & Karasudani, T. (2010), ‘A shrouded wind turbine generating high output power with wind-lens technology.’, *Energies* **3**, 634–649.
- Ohya, Y., Karasudani, T., Sakurai, A., Abe, K. & Inoue, M. (2008), ‘Development of a shrouded wind turbine with a flanged diffuser’, *Journal of wind engineering and industrial aerodynamics* **96**, 524–539.
- Orosa, J., Garcia-Bustelo, E. & Oliveira, A. (2009), Low speed wind concentrator to improve wind farm power generation, in ‘Industrial Electronics’.
- Palma, J. & F.A. Castro, and L.F. Ribeiro, a. A. R. a. A. P. (2008), ‘Linear and nonlinear models in wind resource assessment and windturbine micro-siting in complex terrain’, *Journal of Wind Engineering and Industrial Aerodynamics* **96**, 2308–2326.
- Princeton (n.d.), ‘Streamlines and streamtubes’.
- URL:** <http://www.princeton.edu>

- Saravanamuttoo, H., Rogers, C. & Cohen, H. (2001), *Gas Turbine Theory*, Pearson Education.
- Shames, I. (2003), *Mechanics of Fluids*, McGraw-Hill.
- Shikha, S., Bhatti, T. & Kothhari, D. (2005), ‘Air concentrating nozzles: a promising option for wind turbines’, *Int. J. Energy Technology and Policy* **3**, 394–412.
- Star (2014), *Star CCM+ online manual*, Cd-adapco.
- Stern, F., Wilson, R., Coleman, H. & Paterson, E. (1999), Verification and validation of cfd simulations, Technical report, University of Iowa.
- Versteeg, H. & Malalasekera, W. (2007), *An Introduction to computational fluid dynamics. The fine volume method.*, Pearson Education.
- Wang, F., Bai, L., Fletcher, J., Whiteford, J. & Cullen, D. (2007), Wind tunnel tests on a wind turbine with concentrator and diffuser arrangement, Technical report, Powereng.
- Wang, F., Bai, L., Fletcher, J., Whiteford, J. & Cullen, D. (2008a), ‘Development of small domestic wind turbine with scoop and prediction of its annual power output’, *ScienceDirect* **96**, 1637–1651.
- Wang, F., Bai, L., Fletcher, J., Whiteford, J. & Cullen, D. (2008b), The methodology for aerodynamic study on a small domestic wind turbine with scoop, Technical report, Journal of wind engineering and industrial aerodynamics.
- Wang, F., Bai, L., Fletcher, J., Whiteford, J. & Cullen, D. (2008c), ‘The methodology for aerodynamic study on a small domestic wind turbine with scoop’, *Journal of Wind Engineering* **96**, 1–24.
- Wang, S. & Chen, s. (2008), ‘Blade number effect for a ducted wind turbine’, *Journal of Mechanical Science and Technology* **22**, 1984–1992.
- Wood, D. (2011), *Small wind turbines*, Springer.
- Wright, A. (2005), Aspects of the aerodynamics and operation of a small horizontal axis turbine, PhD thesis, School of Engineering , University of Newcastle.
- Wright, A. & Wood, D. (2004), ‘The starting and low wind speed behaviour of a small horizontal axis windturbine’, *Journal of Wind Engineering and Industrial Aerodynamics* **92**, 1265–1279.

Appendix A

Validation diagrams and figures

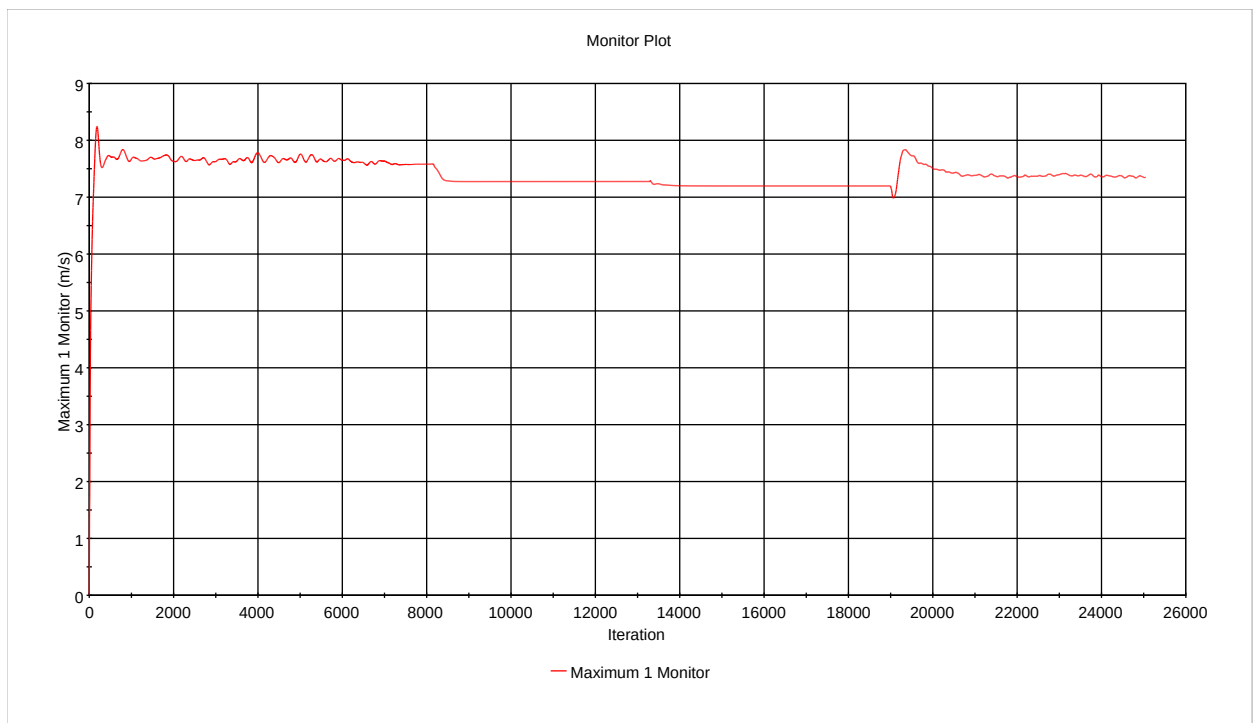


Figure A.1: Monitor Plot of diffuser with brim

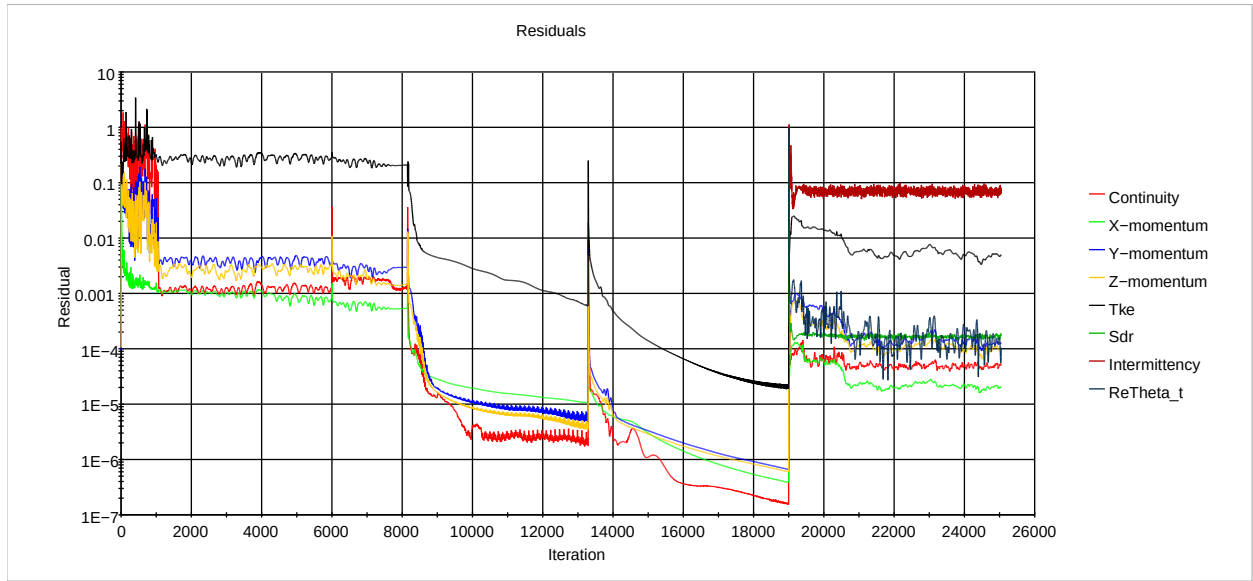


Figure A.2: Residuals for the validation of a diffuser with brim

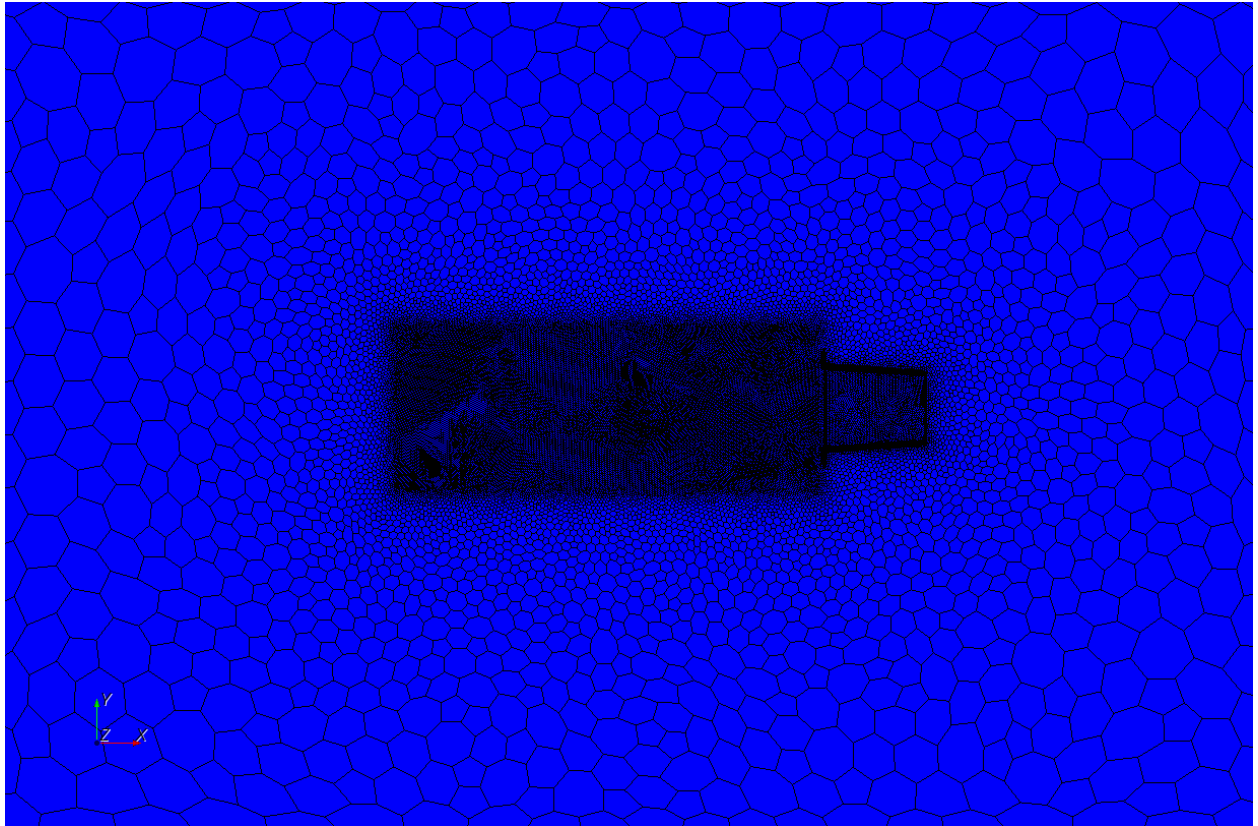


Figure A.3: Plane section through the center of the domain showing the volume mesh of the diffuser model

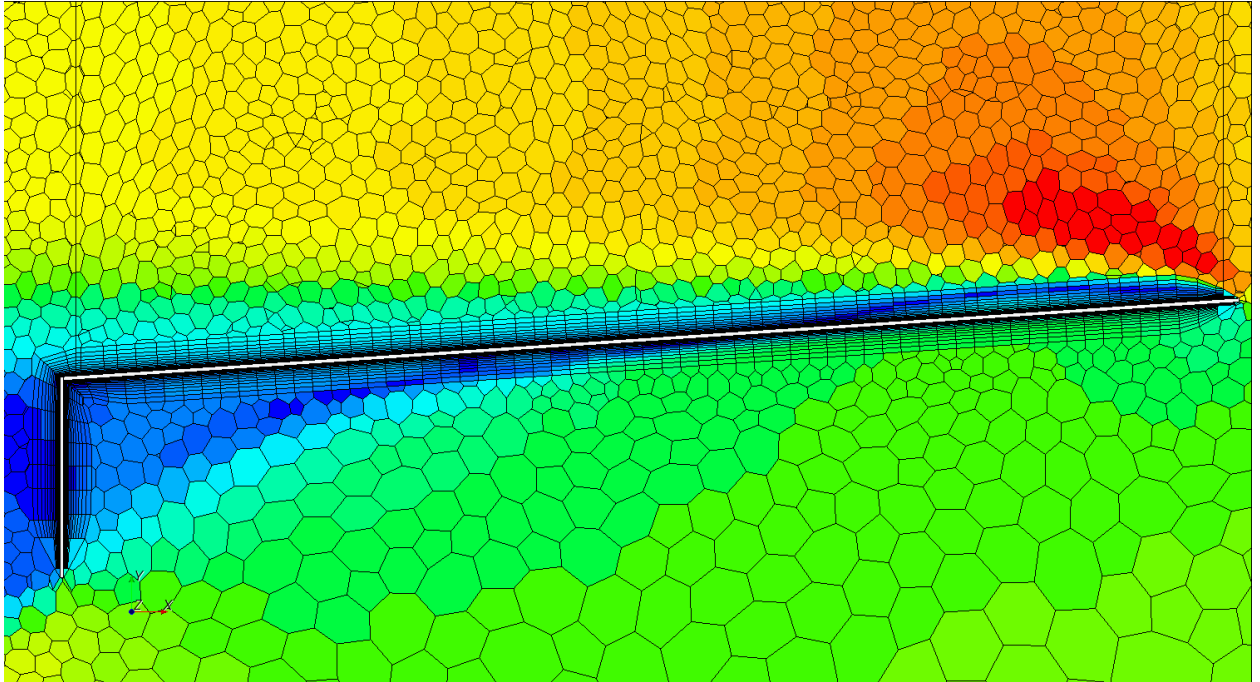


Figure A.4: Plane section through the center showing the mesh at the diffuser wall

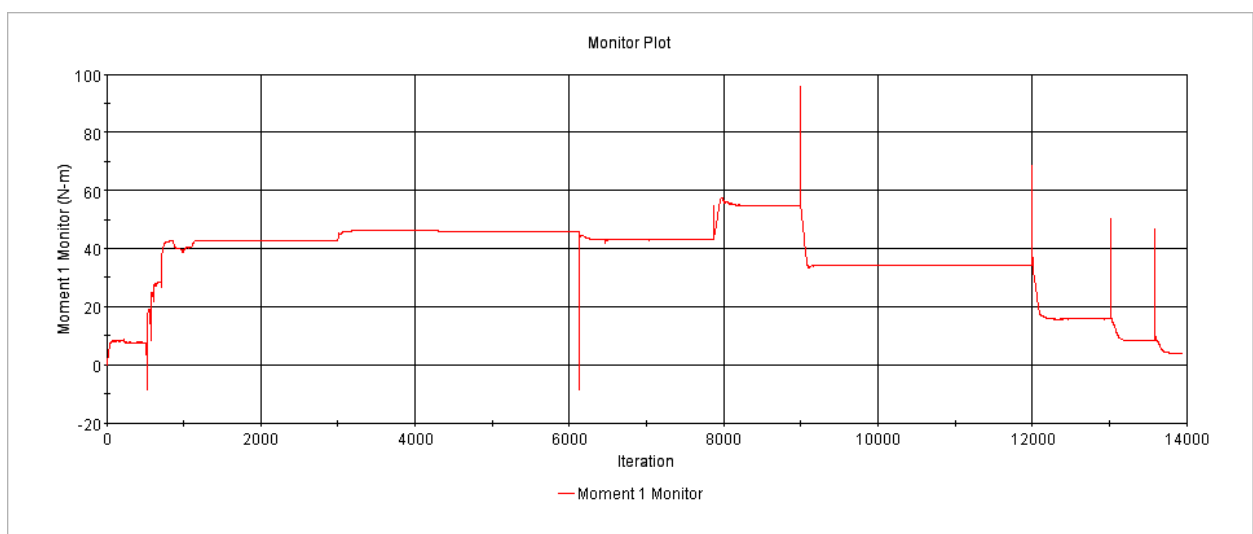


Figure A.5: Momentum monitor plot for the validation of a three bladed open wind turbine

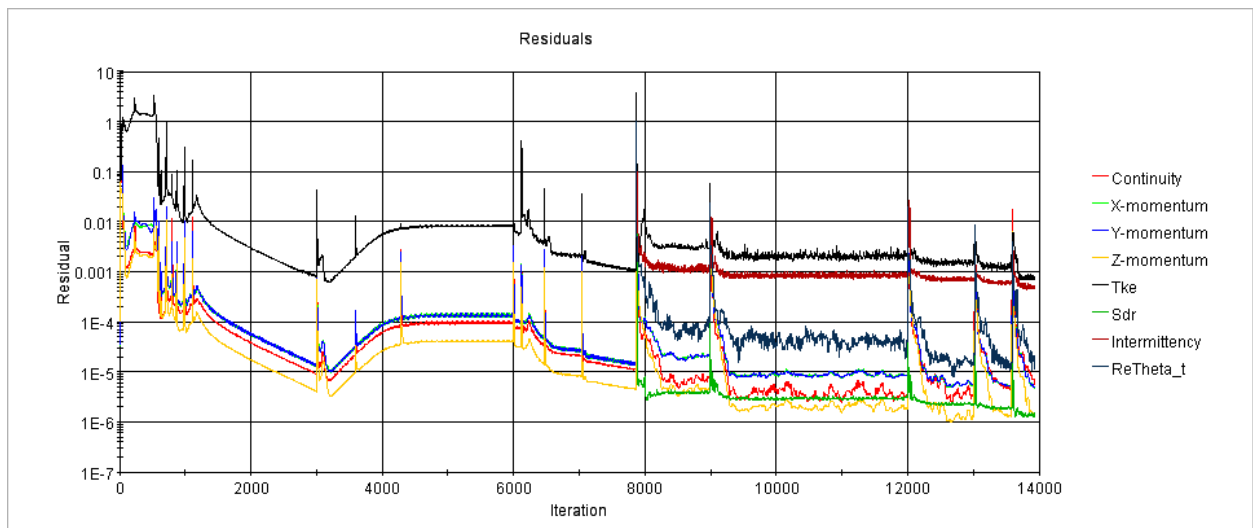


Figure A.6: Residuals for the validation of a three bladed open wind turbine

Appendix B

Design diagrams and figures

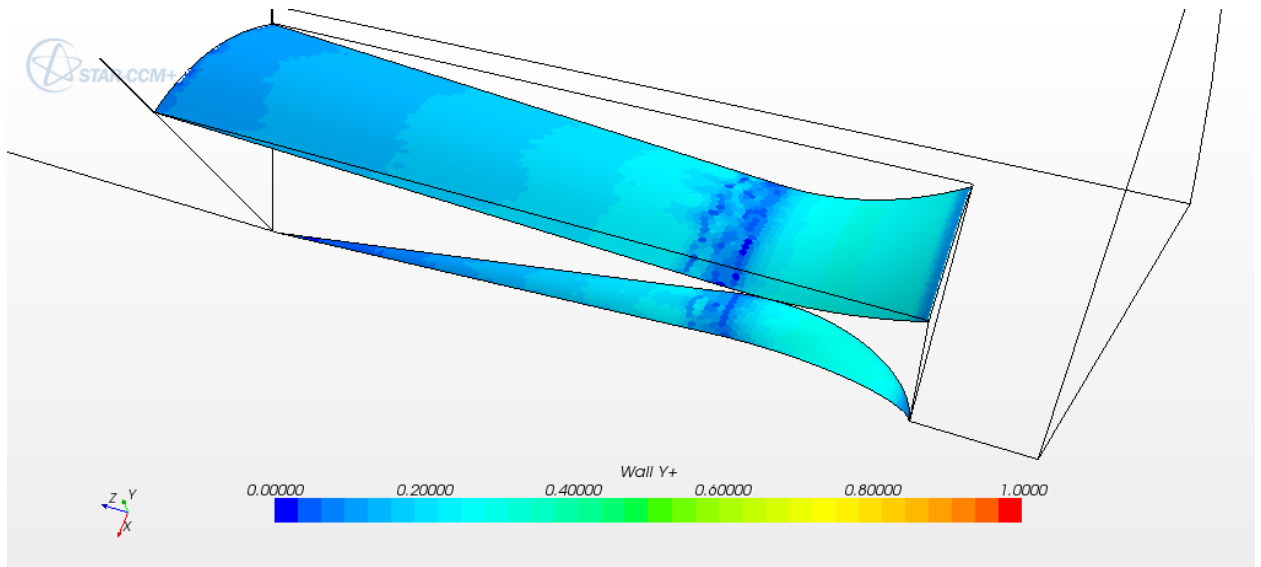


Figure B.1: Wall Y+ values for the inner shroud

Ω (rad/s)	Torque (Nm)	Power (W)	Tip speed ratio	Power Coefficient
15	0.261	3.92	9.25	0.096
10	0.76	6.17	6.17	0.186
5	0.34	3.08	3.08	0.042

Table B.1: Simulation results for an increase of θ_p of 5° on the blade @ a wind speed of 2 m/s

Ω (rad/s)	Torque (Nm)	Power (W)	Tip speed ratio	Power Coefficient
25	1.25	31.25	8.81	0.143
20	3.08	61.6	7.05	0.281
15	2.87	43.05	5.28	0.196

Table B.2: Simulation results for an increase of θ_p of 5° on the blade @ a wind speed of 3.5 m/s

Ω (rad/s)	Torque (Nm)	Power (W)	Tip speed ratio	Power Coefficient
34	3.7	125.8	8.38	0.197
29	6.9	200.1	7.15	0.313
24	7.83	187.9	5.92	0.294

Table B.3: Simulation results for an increase of θ_p of 5° on the blade @ a wind speed of 5m/s

Ω (rad/s)	Torque (Nm)	Power (W)	Tip speed ratio	Power Coefficient
40	10.6	424	7.05	0.242
35	14.59	510.3	6.17	0.0291
30	15.48	464.4	5.28	0.265

Table B.4: Simulation results for an increase of θ_p of 5° on the blade @ a wind speed of 7m/s

Ω (rad/s)	Torque (Nm)	Power (W)	Tip speed ratio	Power Coefficient
50	19.01	950.5	6.85	0.255
45	28.51	1282.95	6.17	0.344
40	29.35	1174	5.48	0.314

Table B.5: Simulation results for an increase of θ_p of 5° on the blade @ a wind speed of 9m/s

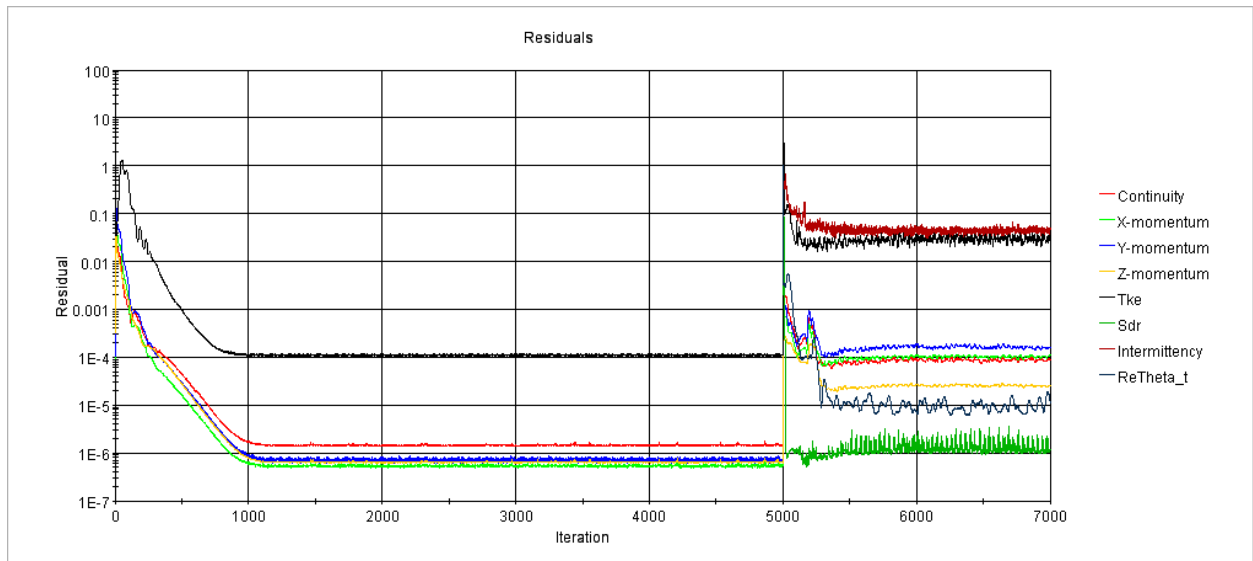


Figure B.2: Residuals of the shroud design

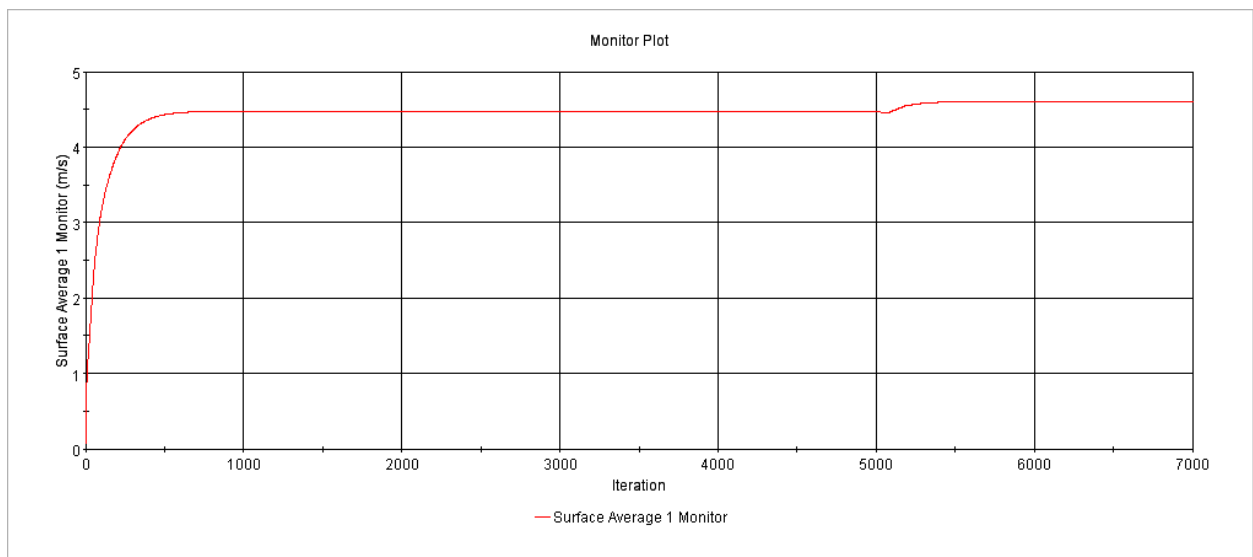


Figure B.3: Monitor plot for the diffuser design

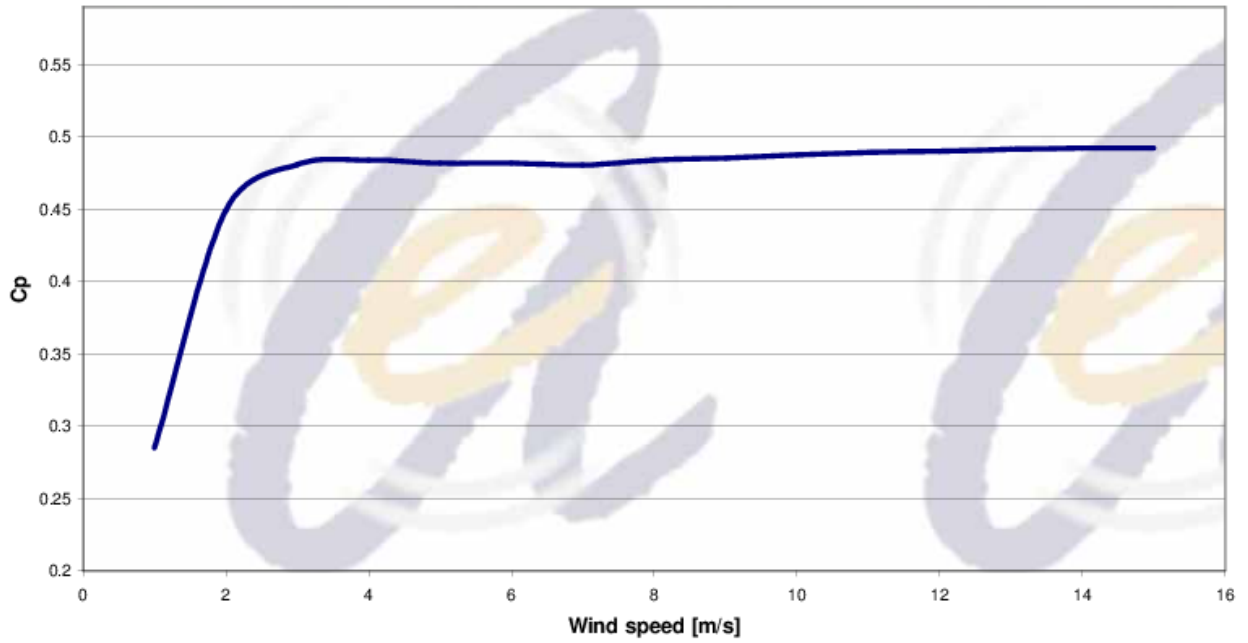


Figure B.4: AE 1.0kW Wind speed/Power Coefficient @ maximum efficiency Bosman et al. (2003)

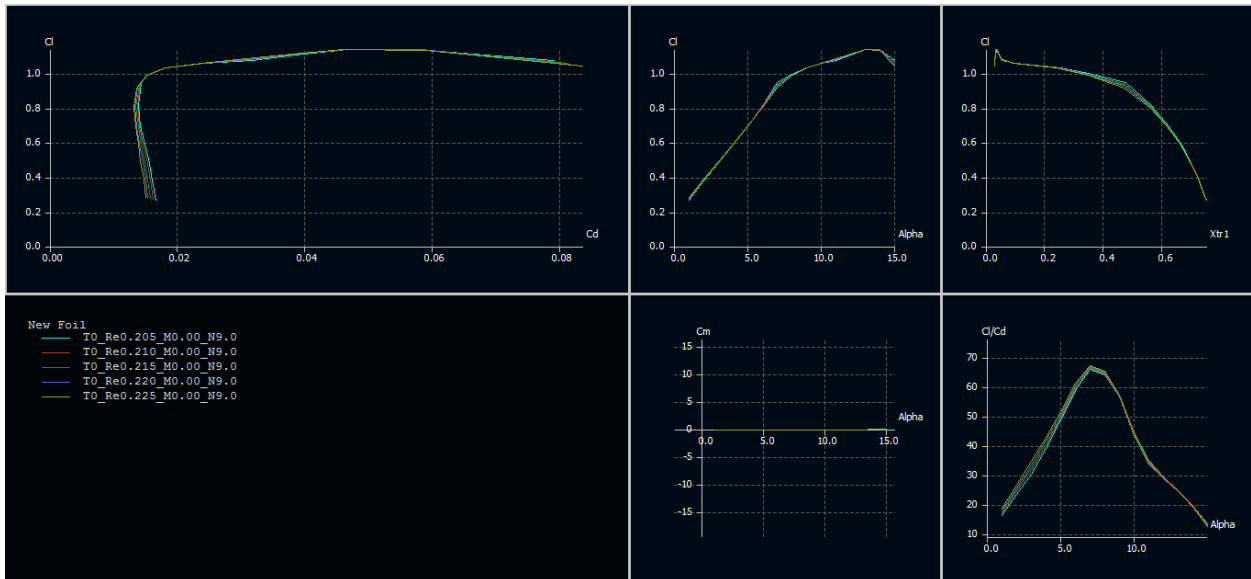


Figure B.5: Two Dimensional airfoil C_l and C_d plots with Re

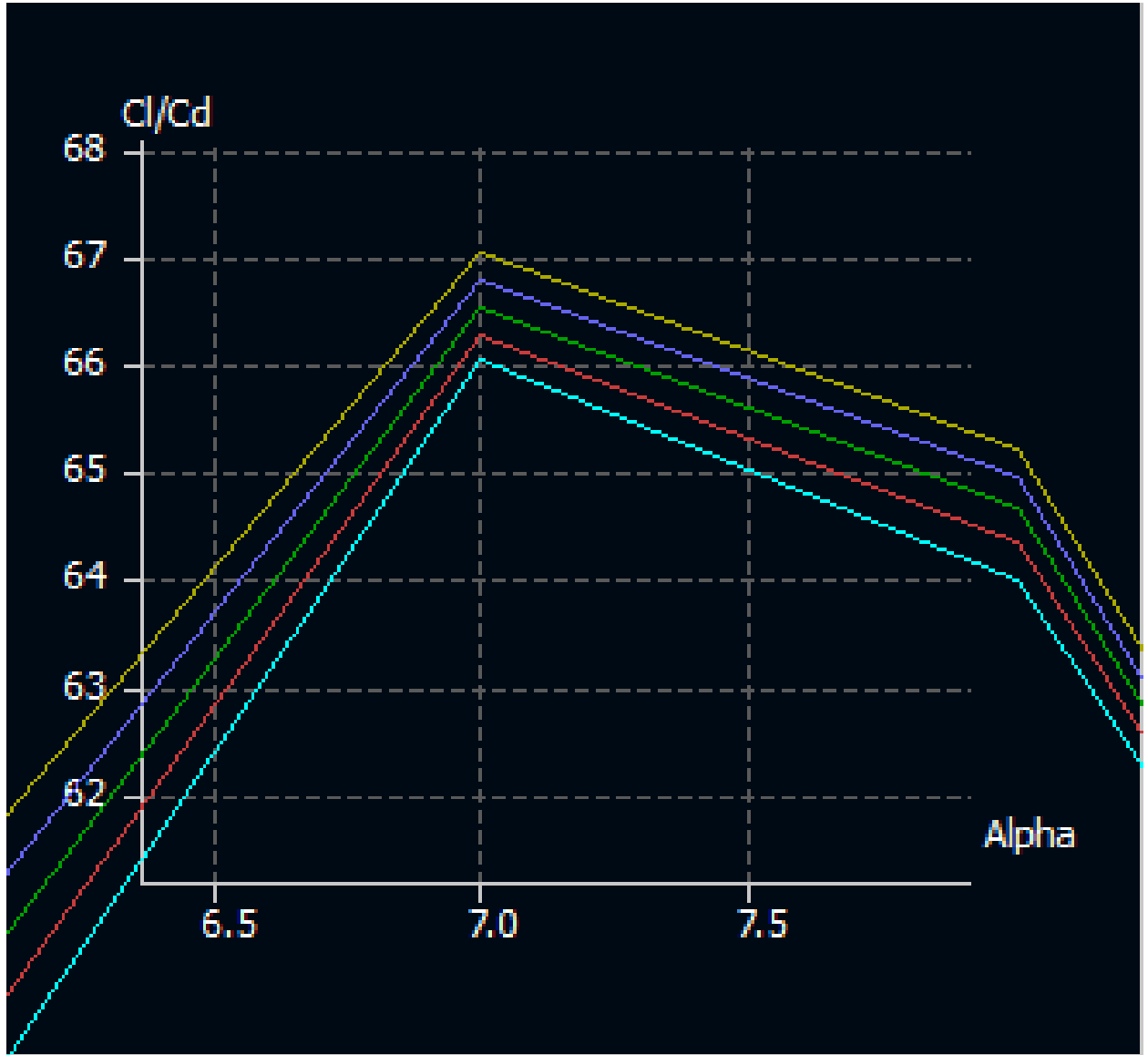


Figure B.6: Two Dimensional airfoil C_l and C_d plots with Re

Maximum Power Coefficient

0.5

RAD/s Ω

32.7

Maximum radius

1.233

Density @T=25 P=86kpa

1.005

Tip speed ratio

5

Velocity in shroud

8.07

Viscosity @T=26

1.85E-05

a

0.333

a'

0.019

Number of blades

3

x=r/R	alfa	Cl/Cd	Radius	Radius in centre	dr	W	c	Drag force	Cd	Cl	Reynolds Number	ϕ Rad	ϕ Degrees	Torque	Available Torque	c/R	
0.543106	2.713452	7	65	0.64	0.66965	0.0593	22.95366	0.15	0.105974	0.015	0.975	186211.41	0.236707	13.5623	0.337571	0.334	0.121654501
0.5912	2.953738	7	65.15	0.6993	0.72895	0.0593	24.87883	0.14	0.115572	0.014919	0.972	188374.05	0.218081	12.49511	0.368416	0.364	0.113544201
0.639294	3.194024	7	65.15	0.7586	0.78825	0.0593	26.81139	0.13	0.124637	0.014919	0.972	188506.26	0.202135	11.58148	0.396253	0.393	0.105433901
0.687388	3.43431	7	65.2	0.8179	0.84755	0.0593	28.74985	0.122	0.134112	0.014877	0.97	189696.14	0.188336	10.79088	0.425293	0.423	0.098945661
0.735483	3.674597	7	65.25	0.8772	0.90685	0.0593	30.69309	0.1145	0.143347	0.014866	0.97	190068.09	0.176283	10.10027	0.453182	0.453	0.092862936
0.783577	3.914883	7	65.25	0.9365	0.96615	0.0593	32.64026	0.108	0.152909	0.014866	0.97	190651.61	0.165666	9.491978	0.481317	0.482	0.087591241
0.831671	4.155169	7	65.25	0.9958	1.02545	0.0593	34.5907	0.103	0.163779	0.014866	0.97	192690.20	0.156246	8.952229	0.513124	0.512	0.083536091
0.879765	4.395455	7	65.25	1.0551	1.08475	0.0593	36.54387	0.098	0.173924	0.014866	0.97	193688.45	0.147832	8.470137	0.542209	0.541	0.079480941
0.927859	4.635742	7	65.25	1.1144	1.14405	0.0593	38.49937	0.0935	0.184171	0.014866	0.97	194683.15	0.140272	8.036991	0.571182	0.571	0.075831306
0.975953	4.876028	7	65.55	1.1737	1.20335	0.0593	40.45686	0.09	0.194867	0.014798	0.97	196923.61	0.133443	7.645739	0.604223	0.601	0.072992701
4.996171				1.233	1.233												
														4.693	4.673		

Figure B.7: Blade design in spreadsheet

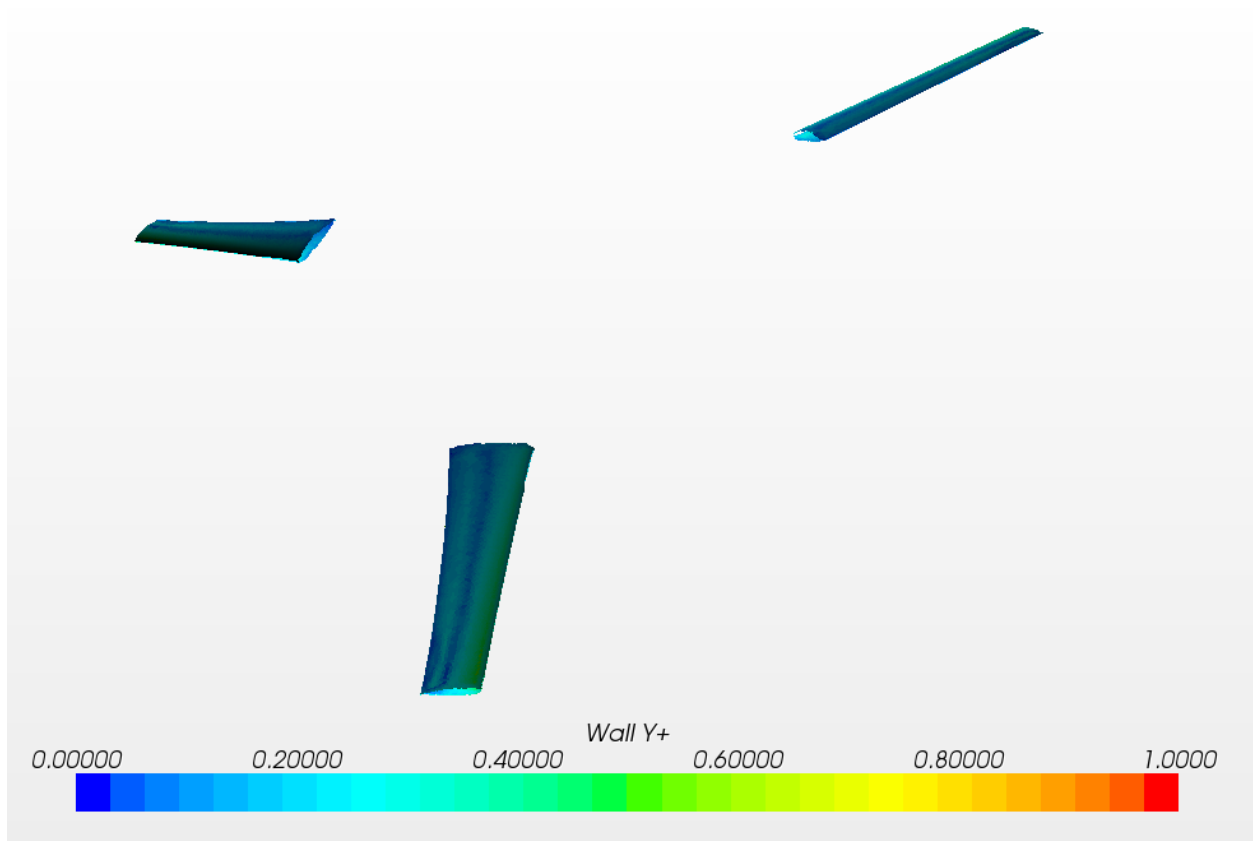


Figure B.8: Wall Y+ values of blades

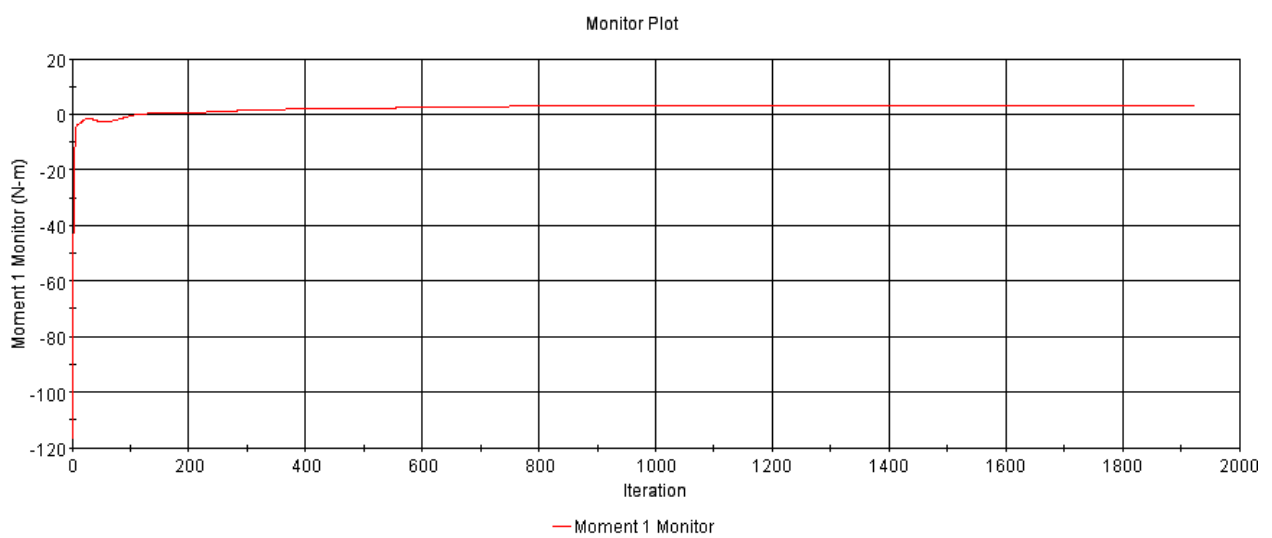


Figure B.9: Monitor plot @ 3.5 m/s wind speed

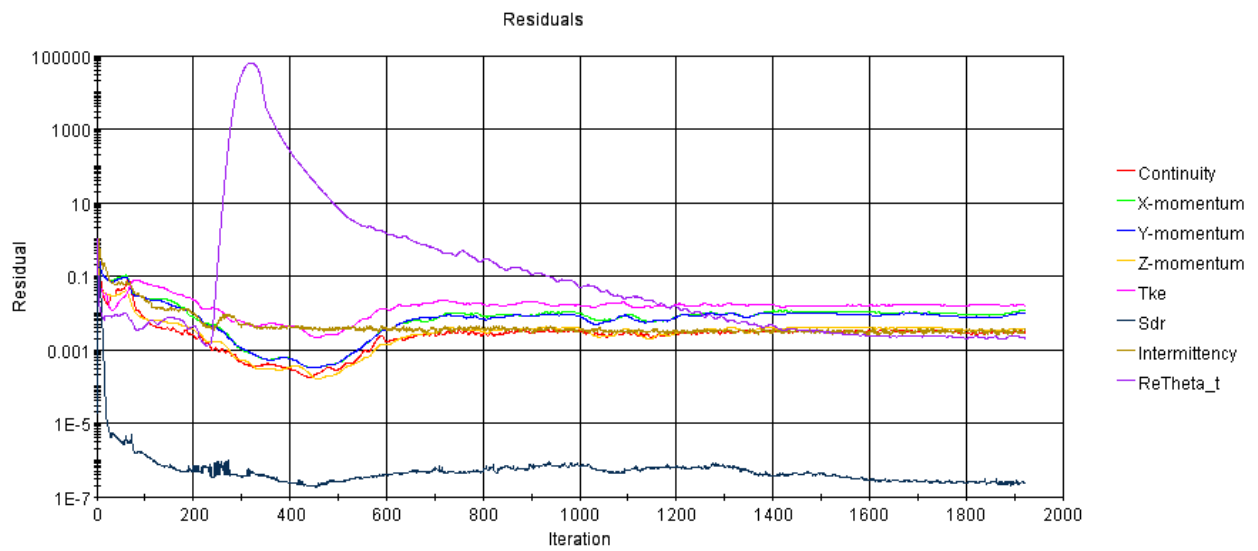


Figure B.10: Residuals plot @ 3.5 m/s wind speed

Appendix C

Results and supportive theory

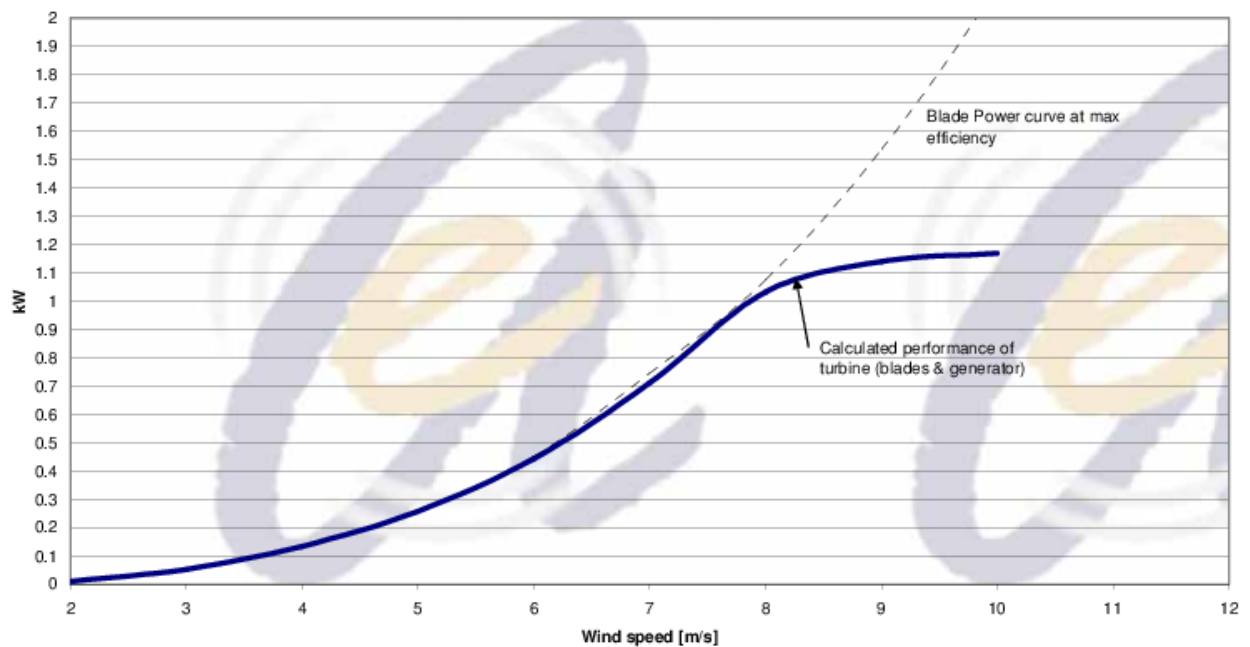


Figure C.1: Wind speed/Power *AE 1.0kW* wind turbine (Bosman et al. 2003)

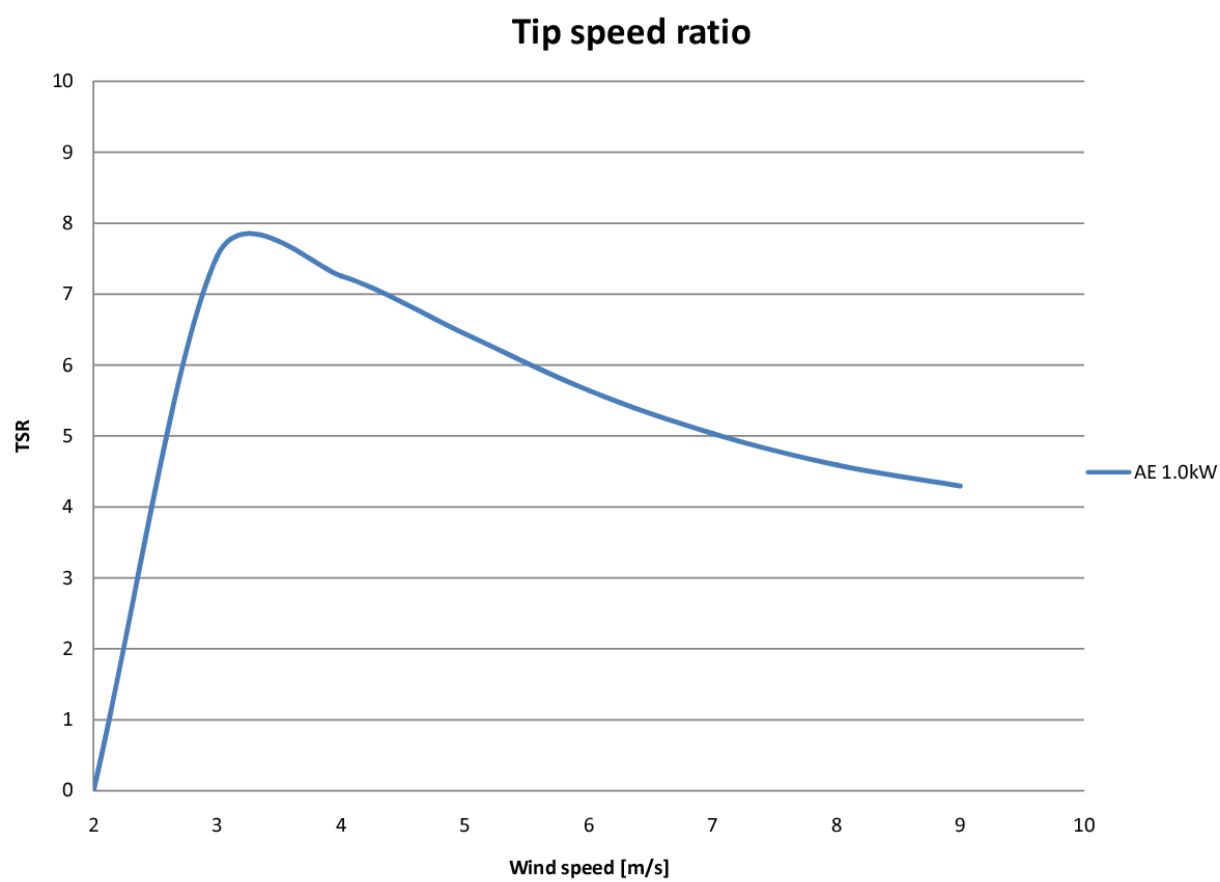


Figure C.2: Tip speed ratio/wind speed *AE 1.0kW* wind turbine