A STUDY OF THE AERODYNAMIC BEHAVIOR OF A NREL PHASE VI WIND TURBINE USING THE CFD METHODOLOGY

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Engineering

By

YEN-PIN CHEN
B.S., Wright State University, 2009

2011
Wright State University
I HEREBY RECOMMEND THAT THE THESIS PREPARED UNDER MY SUPERVISION BY Yen-Pin Chen ENTITLED A Study of the Aerodynamic Behavior of a NREL Phase VI Wind Turbine Using the CFD Methodology BE ACCEPTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF Master of Science in Engineering.

James Menart, Ph.D.
Thesis Director

George Huang, Ph.D.
Chair, Department of Mechanical and Materials Engineering

Committee on Final Examination

James Menart, Ph.D.

George Huang, Ph.D.

Joseph Shang, Ph.D.

Andrew Hsu, Ph.D.
Dean, Graduate School
ABSTRACT

Chen, Yen-Pin, M.S.Egr., Department of Mechanical and Materials Engineering, Wright State University, 2011. A Study of the Aerodynamic Behavior of a NREL Phase VI Wind Turbine Using the CFD Methodology

Wind energy is an abundant natural resource that people have been trying to tap in recent decades. More and more wind turbines are being built to solve the world’s energy shortage problem. For a wind turbine, power extracted from the wind by the rotor and the torque applied to the wind turbine blades are important issues in the design process. Thus there is a need to predict the performance of wind turbine blades using computer modeling. This work shows the results of a computational fluid dynamic simulation developed to predict the air flow field and associated aerodynamic quantities around the moving blades of a wind turbine.

The commercial software package SolidWorks was used to construct the geometrical model. Two commercial CFD codes, SC/Tetra and FLUENT, were used to do the fluid simulations. This work was performed in two phases. First a two-dimensional airfoil simulation was modeled to investigate the aerodynamic coefficients $C_l$, $C_d$, and $C_m$ for the S809 airfoil. Validation of the CFD model was also examined. The second phase of this modeling work was a three-dimensional model of the flow around the NREL (National Renewable Energy Laboratory) Phase VI wind
turbine rotor, which is a horizontal axis wind turbine with two blades using an S809 airfoil. In the three-dimensional model, both rotating blades were simulated. Power extracted from the wind by the rotor, torque on the blades due to the wind, pressure distributions on the blades, and air flow velocity distributions around the blades are the results presented in this work. Comparisons between results obtained from numerical computations and those from the experimental investigation and previous computational investigations are in a good agreement.

Subsequently, using FLUENT codes a detailed study of the effect of yaw angle on power extraction and blade torque was performed. Results are presented for yaw angles of 0°, 10°, 20°, 30°, and 60° and wind speeds of 7m/s, 10m/s, 13m/s, 15m/s, 20m/s, and 25m/s. These results show that yaw angles up to 20° do not cause more than a 2% reduction in power extraction, indicating that wind turbines do not have to be perfectly aligned with the wind for good operation. This is beneficial in practice because it may be difficult to align the wind turbine with the wind direction under the condition of rapidly changing wind directions.
CONTENTS

1. Introduction
   1.1 General Overview of Wind Turbine  2
   1.2 Working Concepts of Wind Turbine Rotor  5
      1.2.1 Characteristic of Wind Resource  5
      1.2.2 Aerodynamics of an Airfoil  7
      1.2.3 Limit Aerodynamic Analysis of Wind Turbine Rotor  12
      1.2.4 Blade Element Momentum Theory  22
   1.3 Literature Review on Airfoils and Wind Turbines  23
   1.4 Scope of research  27

2. Modeling for CFD Simulation  30
   2.1 Computational Fluid Dynamics codes  30
   2.2 Geometric Model for S809 Airfoil  35
   2.3 Geometric Model for NREL Phase VI Turbine Rotor  41
   2.4 Simulation Setup  46
      2.4.1 Register Surfaces and Volume and Assign Material Properties  46
      2.4.2 Analysis Conditions  49
      2.4.3 Post Processing  53

3. Two-Dimensional Airfoil Studies  54
   3.1 Grid Size  55
   3.2 Turbulence Model  58
   3.3 Computational Domain Size  61
   3.4 Comparisons to Other Investigations  62
3.5  Original SST Transition Turbulence Model  69
3.6  Airfoil Characteristics  70
4.  Results of Three-Dimensional Wind Turbine Simulation  77
   4.1  No Yaw Case  77
   4.2  Yaw Cases  93
5.  Conclusions and Future Work  97

REFERENCES  103

APPENDICES
A.  S809 AIRFOIL GEOMETRY DATA FOR TWO-DIMENSIONAL SIMULATION MODELING  106
B.  SECTIONAL GEOMETRY DATA OF NREL PHASE VI WIND TURBINE BLADE FOR THREE-DIMENSIONAL SIMULATION MODELING  108
FIGURES

Figure 1.1  Power generating process of a wind turbine.................................3
Figure 1.2  Wind boundary layers[4] .................................................................7
Figure 1.3  Airfoil nomenclature[5] .................................................................8
Figure 1.4  Forces generated on an airfoil section[7]........................................10
Figure 1.5  Flow over an airfoil[8]..................................................................10
Figure 1.6  Schematic of the one dimensional actuator disc model.................13
Figure 1.7  Power and thrust coefficient for a Betz wind turbine[2]...............17
Figure 1.8  Theoretical maximum power coefficient[2] .................................22
Figure 2.1  NREL S809 airfoil profile............................................................36
Figure 2.2  Model construction in SolidWorks. ...............................................37
Figure 2.3  10m x 10m domain mesh..............................................................40
Figure 2.4  50m x 50m domain mesh..............................................................40
Figure 2.5  200m x 200m domain mesh..........................................................40
Figure 2.6  Mesh around leading edge of S9809 airfoil.................................41
Figure 2.7  Upwind view of NREL phase VI wind turbine in wind tunnel test[2] .............................................................................................42
Figure 2.8  Rotor blade modeling in SolidWorks.............................................44
Figure 2.9  Mesh around cross sectional airfoil generated by SCTpre .........44
Figure 2.10 Mesh around cross sectional airfoil generated by ICEM CFD......45
Figure 2.11 Three-dimensional wind turbine model setup and blade segment in ICEM CFD........................................................................46
Figure 2.12 Unstructured mesh construction in SCTpre.................................47
Figure 2.13 Face registration of two-dimensional airfoil model..................47
Figure 2.14 Face registration of three-dimensional wind turbine model .......49
Figure 2.15  Mesh partition in FLUENT for parallel calculation ......................53

Figure 3.1  Lift coefficient as a function of angle of attack (AoA)..................57

Figure 3.2  Drag coefficient as a function of angle of attack (AoA)............57

Figure 3.3  Moment coefficient as a function of angle of attack (AoA)........58

Figure 3.4  Lift coefficients for different turbulence model..........................59

Figure 3.5  Drag coefficients for different turbulence model........................59

Figure 3.6  Moment coefficients for different turbulence models..................60

Figure 3.7  Lift coefficient for different domain sizes ..............................63

Figure 3.8  Drag coefficient for different domain sizes ..............................64

Figure 3.9  Moment coefficients for different domain sizes ........................64

Figure 3.10 Comparison of lift coefficients to other investigators................65

Figure 3.11 Comparison of drag coefficients............................................66

Figure 3.12 Comparison of moment coefficients........................................67

Figure 3.13 Lift coefficient calculated with original SST transition model.....71

Figure 3.14 Drag coefficient calculated with original SST transition model....71

Figure 3.15 Moment coefficient calculated with original SST transition model...72

Figure 3.16 Pressure distribution around S809 airfoil for angles of attack
from -1.04° to 5.13° ...........................................................................72

Figure 3.17 Velocity contour around S809 airfoil at 2.05° ..........................73

Figure 3.18 Pressure distribution around S809 airfoil for angles of attack
from 6.16° to 12.23° ...........................................................................74

Figure 3.19 Velocity contour around S809 airfoil at 8.20° ..........................75

Figure 3.20 Pressure distribution around S809 airfoil for angles of attack
from 13.22° to 20.16° .......................................................................75

Figure 3.21 Velocity contour around S809 airfoil at an angle of attack ........76
| Figure 4.1 | Shaft torques calculated by SC/Tetra ............................................. 79 |
| Figure 4.2 | Velocity field around the rotor at 7 m/s obtained from SC/Tetra ... 80 |
| Figure 4.3 | Torques acquired in a hand calculation using SC/Tetra produced pressure values ......................................................... 82 |
| Figure 4.4 | Shaft torque calculated by FLUENT .................................................. 84 |
| Figure 4.5 | Power output of turbine as a function of the blade tip speed ratio ... 84 |
| Figure 4.6 | Pressure coefficients at 10m/s and 15 m/s ................................. 87 |
| Figure 4.7 | Pressure coefficients at 10m/s and 15 m/s ................................. 88 |
| Figure 4.8 | Pressure coefficients at 20m/s and 25 m/s ................................. 89 |
| Figure 4.9 | Flow field around the wind turbine at 7m/s .............................. 90 |
| Figure 4.10 | Pressure contours in Pa at 7m/s. ...................................................... 90 |
| Figure 4.11 | Pressure contours in Pa at 10m/s. ..................................................... 91 |
| Figure 4.12 | Pressure contours in Pa at 13m/s. .................................................... 91 |
| Figure 4.13 | Pressure contours in Pa at 15m/s. .................................................... 92 |
| Figure 4.14 | Pressure contours in Pa at 20m/s. .................................................... 92 |
| Figure 4.15 | Pressure contours in Pa at 25m/s. .................................................... 93 |
| Figure 4.16 | Yaw angle ......................................................................................... 94 |
| Figure 4.17 | Calculated shaft torque changes with yaw angle .......................... 95 |
| Figure 4.18 | Comparison of power output with yaw angle ............................... 96 |
TABLES

Table 3.1 Percentage error between calculated aerodynamic coefficients........ 68
Table 4.1 Comparisons of power coefficients................................. 83
Table 4.2 Power coefficients for different yaw angles and wind speeds........ 96
Table A.1 Airfoil coordinates......................................................... 107
Table B.1 Blade geometry data....................................................... 109
# NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>sweep area of wind turbine rotor ($m^2$)</td>
</tr>
<tr>
<td>$a$</td>
<td>axial induction factor</td>
</tr>
<tr>
<td>$a'$</td>
<td>angular induction factor</td>
</tr>
<tr>
<td>$C$</td>
<td>constant</td>
</tr>
<tr>
<td>$c$</td>
<td>chord length (m)</td>
</tr>
<tr>
<td>$C_d$</td>
<td>drag coefficient</td>
</tr>
<tr>
<td>$C_l$</td>
<td>lift coefficient</td>
</tr>
<tr>
<td>$C_m$</td>
<td>moment coefficient</td>
</tr>
<tr>
<td>$C_p$</td>
<td>pressure coefficient</td>
</tr>
<tr>
<td>$C_T$</td>
<td>thrust coefficient</td>
</tr>
<tr>
<td>$D$</td>
<td>drag force (N)</td>
</tr>
<tr>
<td>$D_{oo}$</td>
<td>cross-diffusion term</td>
</tr>
<tr>
<td>$G_k$</td>
<td>generation of turbulence kinetic energy due to mean velocity gradient</td>
</tr>
<tr>
<td>$G_b$</td>
<td>generation of turbulence kinetic energy due to buoyancy</td>
</tr>
<tr>
<td>$G_{oo}$</td>
<td>generation of specific dissipation rate</td>
</tr>
<tr>
<td>$k$</td>
<td>kinetic energy</td>
</tr>
<tr>
<td>$L$</td>
<td>lift force (N)</td>
</tr>
<tr>
<td>$l$</td>
<td>unit length (m)</td>
</tr>
<tr>
<td>$M$</td>
<td>pitching moment (Nm)</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>mass flow rate through control volume (kg/s)</td>
</tr>
<tr>
<td>$P_{rotor}$</td>
<td>power extracted from wind by wind turbine rotor (kW)</td>
</tr>
<tr>
<td>$P_w$</td>
<td>power in wind (kW)</td>
</tr>
<tr>
<td>$p$</td>
<td>pressure at the surface of airfoil (Pa)</td>
</tr>
</tbody>
</table>
\( p_{\infty} \) pressure in freestream (Pa)
\( Q \) shaft torque (Nm)
\( r \) radius (m)
\( S \) user-defined source term
\( T \) thrust worked on wind turbine rotor (N)
\( U \) wind speed (m/s)
\( \tilde{V} \) fluid velocity (m/s)
\( x \) x-direction location (m)
\( Y \) dissipation
\( Y_M \) contribution of the fluctuating dilatation
\( y \) y-direction location (m)
\( y^+ \) dimensionless wall distance
\( \alpha \) angle of attack (degree)
\( \Gamma \) effective diffusivity
\( \varepsilon \) turbulent dissipation
\( \eta \) power coefficient, efficiency of wind turbine rotor
\( \eta_{\text{overall}} \) efficiency of wind turbine
\( \eta_{\text{mech}} \) mechanical efficiency
\( \lambda \) tip speed ratio
\( \lambda_r \) local tip speed ratio
\( \mu \) fluid viscosity (m\(^2\)/s)
\( \mu_t \) eddy viscosity (m\(^2\)/s)
\( \rho \) fluid density (kg/m\(^3\))
\( \sigma \) Prandtl number
\( \Omega \) rotational speed of wind turbine rotor (rpm)
\( \omega \)  specific dissipation rate

\( \omega_{flow} \)  angular velocity imparted to flow stream (rpm)

**Abbreviations**

2D  Two-dimensional
3D  Three-dimensional
AoA  Angle of Attack
BEM  Blade Element Momentum Theory
CFD  Computational Fluid Dynamic
GUI  Graphical User Interface
HAWT  Horizontal Axis Wind Turbine
MRF  Multiple Reference Frames
NREL  National Renewable Energy Laboratory
SST  Shear Stress Transport
TSR  Tip Speed Ratio
ACKNOWLEDGEMENT

There are several people I would like to appreciate their help. Without them, this thesis will never be done. First I want to thank my family. I was alone in the United States during the research process therefore their encouragement is meaningful even it is through a phone call. I would also like to thank for the support and the company from my roommate Zach Gaston and other friends.

I would like to thank my advisor, Dr. James Menart, for leading me through this work. He offered a great assistance on the research and the matters of my international Master’s program. If it were not for his guidance, I would never have such a valuable experience at Wright State University. It was my honor to work with him.

Thanks Dr. George Huang and his research team in Center for High Performance Computing for providing the technical support and the computer resource. Without Dr. Hunag’s help, I would not get the chance to get in Wright State University. Special thanks one of the CHPC team members Sunil Vytla who taught me much knowledge about CFD.

I would like to thank Dr. Jyh-Tong Teng and Dr. Cheng-Hsing Hsu from Chung Yuan Christian University for sparing their precious time to serve on my defense committee and their teaching at the time I studied in CYCU.
Fellow graduate students Caleb Barnes and Sunil Moda also provided a good help on the thesis. Lastly I want to thank Wright State University for funding my Master’s program.
Chapter 1

Introduction

Power issues have become a huge concern in the past four decades. According to the US Department of Energy, if the world keeps consuming fossil fuel without the development of new energy sources, oil supplies can only last around 45 more years. Also, gas and the coal can only be provided for about 150 and 400 years, respectively [1]. Besides the foreseen energy crisis, people’s awareness has been rising of environmental issues caused by burning fossil fuels such as global warming and oil spills. Therefore, finding substitute energy sources is a high priority. There is nearly an unlimited supply of energy coming from natural activity on the earth that can be converted into useful power, such as the wind, sun, tides, and geothermal energy.

Wind power is the main topic discussed in this work. The history of when humans started extracting power from the wind can be traced back to the 1st century when the earliest machine powered by wind can be found. The advent of electricity production from the wind came about in the 19th century; however, its application was...
limited due to structural difficulties and the cost of building wind turbines. In the late 1960s, because of the growing concern for environmental issues, the potential of wind energy was once again noticed. Therefore large scale wind turbines for public use started emerging [2]. In the past 30 years, wind turbines have had a great development, they are more reliable, effective, and environmentally friendly. Other areas of technology, such as material science, aerodynamics, and computer analysis, have been implemented to improve wind turbine designs for advanced power generation at a lower cost. When computational fluid dynamics was applied to simulating the wind turbine rotor, a better understanding of how a wind turbine extracts energy from the wind under different conditions was provided.

1.1 General Overview of Wind Turbines

Connecting the rotor and the generator is done with a series of gears, shafts and supporting bearings for the purpose of conveying the torque and controlling the rotational speed. To convert the rotational motion of the rotor into electricity either an induction or synchronous generator is used. Obviously, the main usage of a generator is to transform kinetic energy to electrical energy; however, a generator can also be used to provide initial rotational speed to the rotor to get the blades started. These
parts of the wind turbine are contained in a nacelle mounted on the top of a tower that elevates the rotor far above the ground to tap into higher wind speeds.

The selection of a supporting tower is profoundly influenced by the characteristic of the terrain where the wind turbine is located. The height of the tower is usually 1 to 1.5 times the rotor diameter, but not less than 20m. Many crucial factors need to be taken into account when building the tower of a wind turbine, such as vibration, power fluctuation and noise production.

Another wind turbinesystem is the yaw mechanism. The yaw mechanism is a required system to keep the rotor facing the incoming wind at the proper angle. Adjusting the angle of the rotor to the wind not only extracts the maximum achievable energy, but also prevents the rotor blades from fracture under severe wind conditions.

![Figure 1.1](image)

Figure 1.1 Power generating process of a wind turbine.

Figure 1.1 shows the general process of power production by a wind turbine. First, the incoming wind carries power, $P_w$, to the rotating rotor. The special geometry of the rotating rotor blades is capable of extracting kinetic energy from the wind, $P_{\text{rotor}}$. 
and providing rotating torque to the generator. The generator transforms this mechanical energy, $P_{\text{out}}$, into the electricity.

The relations between the above powers and power coefficient can be written as,

$$P_w = \frac{1}{2} \rho U^3 A,$$  \hspace{1cm} (1.1)

$$dP_{\text{rotor}} = \Omega dQ,$$ \hspace{2cm} (1.2)

$$\eta = \frac{P_{\text{rotor}}}{P_w},$$ \hspace{2cm} (1.3)

and

$$\eta_{\text{overall}} = \frac{P_{\text{out}}}{P_w} = \eta_{\text{mech}} \eta,$$ \hspace{1cm} (1.4)

where $\rho$ is fluid density, $U$ is the fluid velocity, $\Omega$ is the angular speed of the rotor, $\eta$ is power coefficient which also indicates efficiency, $\eta_{\text{overall}}$ is the efficiency of the entire wind turbine and $\eta_{\text{mech}}$ is the mechanical efficiency. The power generated is equal to the value of the rotational speed multiplied by the torque as shown in Equation (1.3). The size of modern horizontal axis wind turbines are typically between 500kW and 5MW. The most powerful wind turbine in the world can even reach 7.5 MW [3].
1.2 Working Concepts of Wind Turbine Rotor

A key component to efficient wind turbine performance is the interaction between the rotor and the wind. The rotor is able to produce torque for the generator while the wind flows over its surfaces. The ability to extract power comes from the sophisticated shaped airfoil of the wind turbine blades. The crucial factors to power production of a wind turbine are the fluid mechanics of air flow through a power-extracting rotor, and the efficiency of the generator. Much of the discussion on the general aerodynamics of wind turbines given in this chapter has been adapted from wind turbine book by Manwell, McGowan, and Rogers [2].

1.2.1 Characteristic of Wind Resource

The sun plays an important role in generating the renewable energy contained in the earth’s wind resource. Solar radiation causes the uneven heating on different regions of the earth’s surface. Air has a convective phenomenon when there are temperature gradients, hot air rises and cold air sinks. The spatial variation in heat transfer makes air move from high to low pressure, and therefore winds are formed due to the pressure differences across the earth’s surface.

The motion of global wind is controlled by four atmospheric forces, which are
the Coriolis force from the rotation of the earth, the pressure force, the inertial force, and the friction force. From a macro perspective, the Coriolis and pressure forces decide the magnitude and direction of the wind which in turn affect the location choice for the installment of a wind turbine. The wind at a certain location can have notable variations during a short term time period. The wind direction can vary by as much as 180°, which is from upwind to downwind for the wind turbine. Thus, in order to have ideal performance, a yaw mechanism is necessary for a horizontal axis wind turbine. The magnitude of the wind may also be an issue. While faster wind speeds mean more power in the wind, too fast of a wind speed may mean lower performance for the wind turbine, or even worst, damage to the wind turbine.

Wind turbines are generally installed within 80 meters of the ground surface. Irregular terrain like mountains, trees, and buildings create a drag force between the wind and the earth’s surface. Because of frictional forces between the moving wind and the earth’s surface, a boundary layer develops as shown in Figure 1.2b for laminar flow and Figure 1.2c for turbulent flow. Wind boundary layers can be as thick as 1 kilometer. While it is not possible to elevate wind turbine blades 1 km in the air, the higher the blades the more energy that can be extracted from the wind.

Modeling the extraction of energy from the wind resource is a complex endeavor.
As already mentioned the changing wind directions and speeds add complexity to the modeling process. Also mentioned was wind turbulence. Modeling turbulence in any situation is difficult to do precisely. In addition to these characteristics of the wind resource, flow separation, blade vibration, vortices, blade rotation, etc. make wind turbine aerodynamics difficult to model.

![Wind boundary layers](image)

**Figure 1.2** Wind boundary layers [4].

1.2.2 Aerodynamics of an Airfoil

Airfoils, structures with well-designed geometric shapes installed on rotor blades, are crucial factors on extracting power from the wind due to the interaction between the airfoil surface and the fluid flow. A few terms used to indicate the structure of an airfoil are shown in Figure 1.3. An airfoil typically consists of an upper surface and a lower surface. The mean camber line is a series of points located halfway between the upper and lower surfaces. The dull and sharp edges of the airfoil on two different ends
of the mean camber line are called the leading edge and the trailing edge, respectively. The straight line connecting the leading and trailing edges is the chord line of the airfoil, its distance is designated the chord length, $c$. The camber is the distance measured from the mean camber line to the chord line and the thickness of the airfoil is measured from the upper to the lower surface; both lengths are perpendicular to the chord line. Lastly, the angle between the relative wind and chord line, $\alpha$, is defined as the angle of attack.

![Airfoil nomenclature](image)

Figure 1.3 Airfoil nomenclature [5].

When air flows past the airfoil surface, a velocity difference shows up between the upper and lower surfaces due its geometric shape. The upper surface, also called the suction side, has a curvature which produces faster moving air. On the contrary, the curvature of the lower surface, also called the pressure side, produces an airflow that has a slower velocity compared to the suction side of the airfoil. According to Bernoulli’s Equation [6],
\[ \frac{1}{2} \rho U^2 + p = \text{const.}, \quad (1.5) \]

when the velocity of the fluid speeds up the pressure drops and when the velocity of the fluid slows down the pressure increases. This is the reason for the pressure differences between the lower and upper sides of the airfoil. This is shown in Equation (1.5), where \( \rho \) is fluid density, \( U \) is the fluid velocity, and \( p \) is the pressure.

This pressure difference on the two sides of the airfoil results in two characteristic forces and a moment as shown in Figure 1.4. The first force is the lift force which is perpendicular to the direction of the oncoming airflow and toward the suction side. The second force is the drag force which is parallel to the direction of the oncoming airflow and acts toward the flow direction. The drag force is a consequence of both the pressure difference and friction. The resultant force of the lift and the drag produces the pitching moment on an airfoil. The pitching moment usually acts at a location a quarter length of the chord line, \( c/4 \), from the leading edge of the airfoil. This is called the aerodynamic center of the airfoil. As the angle of attack rises, more lift and drag are generated by an airfoil. However, for every airfoil, when the angle of attack reaches a certain angle of attack, the lift and drag change rapidly due to flow separation. When this happens, an airfoil is said to be stalled as shown in Figure 1.5. The laminar boundary layer becomes thicker as the flow travels along the surface and...
may transition to turbulence if the flow speed is high enough. If the angle of attack is too large, an adverse pressure gradient develops on the suction surface which can cause the air flow just above the surface to slow down and even reverse directions. This causes the boundary layer to detach from the airfoil surface causing eddies and vortices to form. The advent of separation causes a sudden drop in the lift force and increase in the drag force.

![Forces generated on an airfoil section](image1)

**Figure 1.4** Forces generated on an airfoil section [7].

![Flow over an airfoil](image2)

**Figure 1.5** Flow over an airfoil [8].
To analyze airfoil aerodynamic problems, this work will use four non-dimensionalized coefficients. Three nondimensional, aerodynamic coefficients that apply to the airfoil as a whole are the lift coefficient,

\[ C_l = \frac{L/l}{\frac{1}{2} \rho U^2 c} = \frac{\text{Lift force/unit length}}{\text{Dynamic force/unit length}}, \]  

(1.6)

the drag coefficient,

\[ C_d = \frac{D/l}{\frac{1}{2} \rho U^2 c} = \frac{\text{Drag force/unit length}}{\text{Dynamic force/unit length}}, \]  

(1.7)

and the moment coefficient,

\[ C_m = \frac{M}{\frac{1}{2} \rho U^2 Ac} = \frac{\text{Pitching moment}}{\text{Dynamic moment}}, \]  

(1.8)

where \( c \) is the chord length, \( l \) is the airfoil span, which is the length of the airfoil perpendicular to the cross section, and \( A \) is the projected area equal to \( c \times l \). These three coefficients will be used later this work. Locally defined coefficients like the pressure coefficient,

\[ C_p = \frac{p - p_s}{\frac{1}{2} \rho U^2} = \frac{\text{Static pressure}}{\text{Dynamic pressure}}, \]  

(1.9)
1.2.3 Limit Aerodynamic Analysis of Wind Turbine Rotors

To begin to understand the aerodynamic efficiency of a wind turbine rotor, one must understand the Betz’s limit [9]. The Betz limit is a fundamental limit on the maximum power that can be extracted from a given wind resource. In addition this also puts a limit on the maximum thrust produced by the wind. Betz’s limit is based on fundamental linear momentum theory and was first published in 1920 to analyze the performance of ship propellers. In the one-dimensional momentum model, a uniform “actuator disc” represents the ideal wind turbine rotor that is placed in a control volume. This control volume contains the boundaries of a stream tube that just touch the outer perimeter of the area swept out by the rotor or propeller. The control volume’s upstream and downstream cross-sections are well away from the rotor or propeller. The schematic of the one dimensional actuator disc model is illustrated in Figure 1.6.

The following assumptions are used in Betz’s limit theory:

1. The wind is a homogenous, steady state, irrational, and incompressible flow.
2. The actuator disc represents an infinite number of rotor blades. Also, the rotor is considered massless; therefore no angular momentum is taken account.
3. There is no wake rotation in the flow downstream of the actuator disk.

4. Both the flow and the thrust are uniform and laminar over the disc and there is no friction drag.

5. The pressure at far upstream and far downstream of the tube is equal to the undisturbed ambient static pressure.

Figure 1.6 Schematic of the one dimensional actuator disc model [2].

One can find the thrust, $T$, which the wind exerts on the disc and the power this turbine can extract from the wind by applying conservation of linear momentum to the control volume shown in Figure 1.6. For a one dimensional, incompressible flow, the thrust is equal to the momentum change of the airflow:

$$T = U_1(\rho AU)_2 - U_4(\rho AU)_4 = \dot{m}(U_1 - U_4), \quad (1.10)$$

where $\rho$ is the air density and $A$ is the cross section area of the disc, and $\dot{m}$ is the mass flow through the control volume. By conservation of mass the mass flow is the same at all cross sections of the stream tube. Thus for steady state flow we can write

$$(\rho AU)_1 = (\rho AU)_4 = \dot{m}. \quad (1.11)$$
Because there is no work done between stations 1 and 2 and stations 3 and 4, Bernoulli’s equation can be used to relate these pressures,

\[ p_1 + \frac{1}{2} \rho U_1^2 = p_2 + \frac{1}{2} \rho U_2^2, \quad (1.12) \]

and

\[ p_3 + \frac{1}{2} \rho U_3^2 = p_4 + \frac{1}{2} \rho U_4^2. \quad (1.13) \]

Thrust can be expressed as the pressure difference on the two sides of the actuator disc,

\[ T = A(p_1 - p_3). \quad (1.14) \]

From the assumptions above, the far upstream and far down stream pressure are both equal to the static pressure and thus \( p_1 = p_4 \). Also, the flow velocity does not change across the disc and thus \( U_2 = U_3 \). Equation (1.12) and (1.13) can be simplified to obtain a relation for \( p_2 \) and \( p_3 \). This can then be substituted into Equation (1.14) giving:

\[ T = \frac{1}{2} \rho A(U_1^2 - U_4^2). \quad (1.15) \]

The power output, \( P \), equals the thrust times the velocity in front of the disc

\[ P = \frac{1}{2} \rho A(U_1^2 - U_4^2)U_2. \quad (0.16) \]
To calculate the rotational speed of the actuator relative to the wind speed at which the maximum power and thrust are generated, an axial induction factor, \(a\), is defined as

\[
a = \frac{U_1 - U_2}{U_2}. \tag{1.17}
\]

Combining Equations (1.10), (1.11) and (1.15) gives

\[
U_2 = \frac{U_1 + U_4}{2}. \tag{1.18}
\]

By rewriting the axial induction factor equation (Equation (1.17)) one gets

\[
U_2 = U_1(1 - a). \tag{1.19}
\]

By substituting Equation (1.19) into (1.18) one can obtain

\[
U_4 = U_1(1 - 2a). \tag{1.20}
\]

Substituting the new \(U_2\) and \(U_4\) into Eqns. (1.15) and (1.16), the axial thrust and the power output can be written in terms of the axial induction factor as,

\[
T = \frac{1}{2} \rho A U^2 [4a(1 - a)], \tag{1.21}
\]

and

\[
P = \frac{1}{2} \rho A U^3 4a(1 - a)^2. \tag{1.22}
\]
The power output of the rotor can be written as a ratio between the actual power extracted by the rotor to the power in the wind giving the power coefficient, $\eta$, 

$$
\eta = \frac{P}{\frac{1}{2} \rho A U^3 A} = 4a(1-a)^2 \quad (1.23)
$$

The power coefficient is many times used to describe the performance of a wind turbine and can be recognized as the efficiency of the power extraction by the rotor. In this work the both the term power coefficient and rotor efficiency are used to refer to this quantity.

To get the maximum power, take the differential of $\eta$ and set $d\eta = 0$ obtaining $a = 1/3$. Substitute $a = 1/3$ back into Equation (1.23) to obtain a maximum power extraction efficiency of $\eta_{max} = 12/67 = 0.5926$. This result indicates that the most power a rotor can extract from the wind is 59.26% of power in the wind. This is called the Betz limit.

Similar to the power production, the thrust can also be non-dimensionalized as a thrust coefficient, 

$$
C_T = \frac{T}{\frac{1}{2} \rho A U^2 A} = 4a(1-a) \quad (1.24)
$$

By setting the derivative of the thrust coefficient equal to zero, the highest thrust
coefficient can be found at $a = 1/2$ to be 1.0. At the maximum thrust operating point the wind speed far downstream of the wind turbine is zero. At the Betz power limit, $a = 1/3$, and the thrust coefficient has a value of $8/9$. Figure 1.7 is a diagram that shows the variations of power and thrust coefficient with the change in axial induction factor.

![Figure 1.7](image)

Figure 1.7  Power and thrust coefficient for a Betz wind turbine [2].

Although the Betz limit has proven that an ideal turbine can extract as much as 59.26% of the wind’s energy, in practice there are three main effects that reduce the total amount of power production. These effects are:

1. rotation of the wake behind the rotor,
2. a finite number of blades and tip losses, and
3. viscous flow that creates non-zero aerodynamic drag.

The spinning rotor blades generate angular momentum, which causes the rotation of the wake, which is related to rotor torque. The previous theory using linear
momentum coupled with angular momentum can be applied to the case with rotation of the wake. When wake rotation is included, it can be observed that the power extracted is lower than when no wake rotation is included in the model. The power output is reduced by the rotational kinetic energy contained in the wake. Since wake rotation is the result of the torque output from the rotor, high torque tends to create more kinetic energy in the wake, thus more energy loss when the wind flows through the rotor. Based on this finding, wind turbines that operate at high rotational speed with low torque have lower losses than turbines that operate at slow rotational speeds with high torque.

A more detailed annular stream tube model is used when rotation of the wake is taken into consideration in the analysis. The stream tube has a radius \( r \), thickness \( dr \), and a cross-sectional area \( dA = 2\pi r dr \). The rotation of the wake, pressure, and induction factor are assumed to be functions of rotor radius. The pressure difference before and after the blades can be written as

\[
p_2 - p_3 = \rho (\Omega + \frac{1}{2} \Omega_{\text{flow}}^2) \omega_{\text{flow}} r^2,
\]

where \( \omega_{\text{flow}} \) is the angular velocity imparted to the flow stream and \( \Omega \) is the angular velocity of the rotor, both in units of rad/s. Equation (1.14) shows that the thrust is the pressure difference across the rotor which can also be applied to the wake rotation.
case. Multiplying Equation (1.25) by an annular differential area provides an expression for the differential thrust, \(dT\), on this differential area, \(dA\),

\[
dT = (p_2 - p_3)dA = (\rho(\Omega + \frac{1}{2} \omega_{flow})\omega_{flow} r^2)2\pi rdr.
\]

(1.26)

This can be written in terms of the angular induction factor which is customarily defined as

\[
a' = \frac{\omega_{flow}}{2\Omega},
\]

(1.27)
giving

\[
dT = 4a'(1 + a') \frac{1}{2} \rho \Omega^2 r^2 2\pi rdr.
\]

(0.28)

The thrust on an annular element can also be expressed using the axial induction factor as

\[
dT = \frac{1}{2} \rho U^2 4a(1 - a)2\pi r dr.
\]

(1.29)

Setting Equation (1.28) equal to Equation (1.29) and simplifying gives two new quantities defined as local tip speed ratio \(\lambda_r\) and tip speed ratio \(\lambda\) (TSR),

\[
\sqrt{\frac{a(1 - a)}{a'(1 + a)}} \frac{r}{\Omega r} = \frac{\lambda r}{R} = \lambda = \frac{\lambda}{R},
\]

(1.30)

and
\[ \frac{\Omega R}{U} = \lambda \]  

(1.31)

where \( r \) is any given radius on the rotor and \( R \) is the full radius of the rotor. The tip speed ratio is the ratio of the speed at the blade tip to the free stream wind speed. Higher TSRs cause the blades to have large centrifugal forces and to produce more noise. Thus the TSR is an important parameter in designing rotor blades.

By applying conservation of angular momentum, an expression for torque can be developed. Note that the torque delivered to the rotor, \( Q \), must equal the change in angular momentum of the air in the wake, which gives

\[ dQ = d(\omega_{\text{flow}})r = 4a'(1 - a)\frac{1}{2} \rho U \Omega r^2 2\pi r dr \]  

(1.32)

It was mentioned previously that the power generated is equal to the value of the rotational speed multiplied by the torque. Substituting the equation for the tip speed ratio, Equation (1.31), and the torque exerted on the rotor, Equation (1.32), into the equation for power generated yields

\[ dP = \frac{1}{2} \rho AU^3 [\frac{8}{\lambda^2} a'(1 - a)^3 \lambda r d\lambda] \]  

(1.33)

Similar to the previous analysis, the power coefficient is the ratio of power generated to the power in the wind. The only difference is that the differential power is used in the power coefficient on an annular element instead of total power.
\( d\eta = \frac{dP}{\frac{1}{2} \rho AU^3} \) \hspace{1cm} (1.34)

After integration, algebraic manipulations, and variables changes, the power coefficient can be presented in a form to predict maximum theoretical efficiency considering wake rotation. The result is [2]

\[
\eta_{\text{max}} = \frac{8}{729\lambda^2} \left[ \frac{64}{5} x^5 + 72x^4 + 124x^3 + 28x^2 - 63x - 12\ln(x) - 4x^{-1} \right]_{x=1-3\lambda} \quad (1.35)
\]

The result of this analysis is graphically illustrated in Figure 1.8 to compare with the Betz limit. It can be seen that the power coefficient cannot reach the Betz limit due to the loss caused by fluid rotation in the wake. Also, the power coefficient increases with the tip speed ratio. Thus, as mentioned above, the statement that turbines with high rotational speed and low torque have less loss than turbines with low rotational speed and high torque is proven.

Another effect influencing the total amount of power that can be produced is the number of rotor blades. Note that the number of blades is assumed to be infinite in the Betz limit development that provided a maximum achievable power coefficient of 59.26%. Comparing Betz’s limit with the results of turbines using one, two and three blades, the more blades on a turbine, the closer the power coefficient gets to the Betz limit. However, the increase in power coefficient going from one to two blades is
greater than the increase in power coefficient going from two to three blades and so on. Thus, in practice, considering the increased efficiency with more blades and the increased cost of more blades, most wind turbines are installed with two or three blades. Generally, two-bladed turbines tend to use a higher tip speed ratio to make up for the power output difference compared to three-bladed turbines.

![Diagram](image.png)

**Figure 1.8** Theoretical maximum power coefficient [2].

### 1.2.4 Blade Element Momentum Theory (BEM)

Blade element momentum theory (BEM) is commonly used to design the optimum shape of the rotor blades [10]. The momentum theory previously used to predict efficiency is a control volume analysis that finds the forces exerted on a wind turbine by applying the conservation of linear and angular momentum. Nevertheless,
the momentum theory does not provide enough equations if one looks closer at the flow condition at the rotor blades. The flow condition is controlled by the characteristics of the airfoil installed on the blades and the twist distribution of the blades. Blade element theory was developed to analyze the forces exerted on different sections of the blades. This is an approximate analytical technique used to design wind turbine rotors. This technique is not used in this work, where a detailed computational fluid dynamics (CFD) analysis is done. If the reader is interested in BEM a detailed discussion can be found in Manwell, McGowan, and Rogers [2]. A literature review on some of the CFD analysis that has been done on wind turbine rotors and airfoils is presented in the next section.

1.3 Literature Review on Airfoils and Wind Turbines

Computer simulation has become a necessary analysis technique in wind turbine development. This technique helps researchers simulate problems on material mechanics, electric engineering and fluid dynamics, where fluid dynamics is the focus of this thesis. With the progress of computer science, computer simulation is capable of handling sophisticated jobs and providing accurate results. Compared with experimental investigation, simulations are very inexpensive. An experiment is
doubtlessly the best way to ensure the best representation of reality, but these days’ computational techniques for fluid dynamic problems are providing reasonable results. In addition, computer simulation usually provides much more detailed results than experimental techniques. When computer simulation is adopted, less time is required to complete a wind turbine design. Also, the parameters can be easily modified for different cases because of the flexibility of computer simulation.

Numerous experimental and CFD studies have been done on wind turbine rotors and airfoils up to this time. In this section a review of previous works related to this research are presented.

Somers and Tangler [11] conducted a low turbulence wind tunnel test in the laboratory of Delft University. The tested model was an S809 airfoil that is exclusively designed for HAWT (horizontal axis wind turbine) applications. The aerodynamic characteristics of the airfoil were predicted theoretically first, then verified by experimentation. The experiment was run at various angles of attack and Reynolds numbers. The maximum lift coefficient and a low profile drag coefficient were the two main objectives to be examined. The experimental results showed good agreement with the theoretical results. In the experiment the docile stall and insensitivity to roughness were displayed. The results were also compared with the
aerodynamic characteristics of other airfoils.

Wolfe and Ochs [12] developed a computer simulation for an S809 wind turbine airfoil using the commercial code CFD-ACE. The CFD calculation was performed in steady state with two spatial dimensions with the k-ε turbulence model. This work focused on two objectives, transition prediction and turbulence modeling. The simulation results agree well with Somers and Tangler’s [11] wind tunnel test results on the comparisons of the lift coefficient, the drag coefficient, and the pressure distribution. The results indicate that the starting point of the transition needs to be modeled correctly to increase the accuracy of simulating the attached flow. Also, the commonly used turbulence model, the k-ε model, is proven not to be appropriate for solving flow separation at large angles of attack.

Hand et al. [13] conducted a full-scale wind tunnel test in the National Renewable Energy Laboratory’s (NREL) project called the Unsteady Aerodynamic Experiment. This test is the further development of Somers and Tangler’s [11] work. The S809 wind turbine airfoil was installed on the tested model NREL Phase VI wind turbine. The purpose of this work was to acquire valid data of the aerodynamic characteristics and structural behavior. Several important characteristics of the wind turbine rotor were probed throughout the test, such as blade surface pressure, angle of
attack, pressure distribution at five span locations on a blade, shaft bending moment, and yaw moment. The results provided the performance of the tested wind turbine and are valid to be a reference for developing an enhanced wind power machine.

Sezer-Uzol and Long [13] performed a three-dimensional and time-accurate CFD simulation of the NREL Phase VI wind turbine. The simulation was performed using the three-dimensional, unsteady, parallel, finite element flow based commercial code PUMA2. An unstructured moving mesh was adopted for the simulation of the rotation of the wind turbine rotor. Three initial conditions with different wind speeds and yaw angles were examined: 7 m/s with 0 yaw degree, 15 m/s with 0 yaw degree, and 7 m/s with 30 yaw degree. Comparisons of computational results and experimental results were made, which shows good agreement for pressure distributions at different blade spans. The appearance of flow eddies and separation were discussed in the published paper (Sezer-Uzol and Long, 2006). The results of this work are helpful for analyzing the underlying turbulent flow and noise produced.

Hartwanger and Hovart [15] combined both computational and theoretical calculations in their work. The computer simulation was developed using the commercial CFD software ANSYS-CFX. Firstly, the two-dimensional airfoil was simulated and compared with Wolfe and Ochs’s [12] experimental results. The
A three-dimensional model was constructed and validated based on the results of a two-dimensional airfoil simulation. Then the three-dimensional results were used to estimate the induction factor of the actuator disk. Lastly, the estimated induction factors were used to modify the classical actuator disk model. The results obtained from the modified actuator disk model show good agreement with another simulation and experimental results.

1.4 Scope of Research

Although computer simulation has been widely adopted for solving engineering problems, modeling airflow over wind turbine blades is still challenging. The difficulties are caused by the complex geometry due to the blade twist and pitch angles, as well as the flow condition changes caused by various angles of attack and flow separations. However, these challenges are not merely obstacles, but also motivations for researchers to develop more accurate simulations of wind turbines. There are two primary objectives in this work: the first objective is to compare how the aerodynamic characteristics of an S809 airfoil change with different boundary conditions and the second objective is to observe the influence of wind direction on power production of the NREL Phase VI wind turbine.
This research covers computer simulations on both two-dimensional airfoils and a three-dimensional wind turbine rotor carried out by two commercial CFD codes, SC/Tetra and FLUENT. The two-dimensional S809 airfoil simulation is a further analysis of the work performed by Wolfe and Ochs [12]. It has been proven explicitly by Wolfe and Ochs [12] that the $k-\varepsilon$ turbulence model is not suitable for solving flow separation. Thus there is a need to test other available turbulence models to achieve higher accuracy. Also, grid size and computational domain size are taken as parameters in the simulations. All the two-dimensional results are compared with Delft University’s S809 airfoil wind tunnel report by Somers and Tangler[11].

The two-dimensional airfoil information, such as boundary condition setup, model construction, and computational results, are used, as a basis to use the right type of modeling techniques for the three-dimensional wind turbine simulations. In this work there are two key factors that contributed to the completion of the simulations. One is the model construction of the rotor blades with CAD software. The dimensions of the NREL Phase VI wind turbine is provided in the NREL wind turbine report by Hand et al. [13]. The second factor is the CFD model itself. This includes the boundary condition setup; choosing an appropriate turbulence model, and the utilization of a moving mesh for simulating rotor rotation. The choice of the
turbulent model is based on the two-dimensional simulations done as part of this work and the moving mesh is based on the work done by Sezer-Uzol and Long [14] and Hartwanger and Hovart [15].

The first goal of the three-dimensional simulation is to match the calculated power production to experimental values for a yaw angle of zero degrees. After validation of a no yaw simulation, yaw angle changes are applied to the model to visualize their effect on the power production and the wake rotation. Also, the post-processing of the flow field and pressure distribution around the wind turbine rotor is discussed. This research presents the validation and the improvement of the previously performed computational work in this area. The results also provide helpful information for future wind turbine development.
Chapter 2

Modeling for CFD Simulation

This chapter presents the methodology of the simulation used in this research. It consists of three phases: geometric model construction, simulation setup and post-processing. The geometric model construction includes building the model with CAD software and meshing for CFD solvers. The simulation setup is the control of solver parameters, such as the initial condition, boundary conditions and moving mesh, to help the calculation reach convergence fast and easily. The purpose of post-processing is to visualize the results in a user friendly manner. Before getting into these three aspects of the CFD model a section will discuss some aspects of the two commercial CFD codes used to perform this work.

2.1 Computational Fluid Dynamic Codes

SC/Tetra [16] and FLUENT [17] are two commercial CFD (computational fluid
dynamic) codes adopted for this research. SC/Tetra is mainly used to perform the
two-dimensional S809 airfoil simulation. Both SC/Tetra and FLUENT are used to
perform the complex three-dimensional simulations of the wind turbine. The reason
for adding the commercial CFD software FLUENT was SC/Tetra was not producing
good results. Comparisons between the solutions from these two software packages
will be made.

SC/Tetra was developed by the software company CRADLE in 1998, the version
used in this work is 8.0. It features a user-friendly interface and can be used for a wide
range of fluid and thermal applications. SC/Tetra is composed of three main parts:
SCTpre, SCTsolver, and SCTpost. SCTpre deals with the pre-processing and includes
automatic mesh and hybrid mesh generation, and also carries out boundary condition
and material properties setup. SCTsolver plays the role of handling the calculation, as
well as monitoring the progress and convergence status. SCTpost transfers the
numerical results into a user-friendly visualization. A broad variety of graphical
results can be shown in SCTpost.

FLUENT is a CFD computation product of the ANSYS Corporation, the version
used in this work is 13.0. Similar to SC/Tetra, FLUENT contains different tools in the
overall package that perform different aspects of the simulation. Meshing for
FLUENT can be performed by a few different meshing packages, ICEM CFD is one of them and this meshing software is chosen to generate the mesh for the three-dimensional model of the wind turbine simulated in this research. Mesh generation in ICEM CFD is not limited to a certain kind of mesh; it provides flexibility for the user to choose from structured and unstructured meshes. Other than meshing, pre-processing, solving, and post-processing can all be executed in one graphical user interface (GUI) of FLUENT.

The condition of fluid flow over an airfoil or a wind turbine blade can be modeled using the Navier Stokes equations. For a three-dimensional simulation the Navier stokes equations are composed of one equation describing conservation of mass (continuity equation), three equations describing conservation of momentum and one equation describing conservation of energy. In this work there are no heating or cooling issues and fluid properties are taken as being constant. In addition, the flow speeds are too slow for compressibility effects to be noticeable. Thus the conservation of energy equation is not required. For the airfoil and wind turbine simulations done in this work the Navier-Stokes equations simplify to [18],

\[ \text{[18]} \]
\[ \nabla \cdot \vec{V} = 0, \quad (2.1) \]

and

\[ \rho \frac{D\vec{V}}{Dt} = -\nabla \rho + (\mu + \mu_t)\nabla^2 \vec{V}. \quad (2.2) \]

It is not an easy job to solve the incompressible Navier-Stokes equations due to the coupling between the momentum equations and the continuity equation and the need to solve for the unknown pressure field. To solve for the three unknown velocities and the unknown pressures the semi-implicit method for pressure-linked equations called SIMPLE is applied. In SIMPLE, the momentum equation is solved with the old pressure value to get the approximate velocity field. The new pressure term can be obtained by reformulating the momentum and continuity equations. Once the pressure field is determined from the reformulated momentum and continuity equations they are used to calculate and update the velocity field so that it satisfies conservation of mass. The incompressible flow problem can be solved iteratively by repeating this trial and error solution technique. The detailed steps of the SIMPLE algorithm are as follows [19]:

1. Set the boundary conditions.
2. Solve the discretized momentum equation to compute the intermediate velocity field using a guessed pressure field.
3. Compute the mass fluxes at the cell faces.
4. Solve the pressure equation and apply under-relaxation.
5. Correct the mass fluxes at the cell faces.

6. Correct the velocities on the basis of the new pressure field using the momentum equations.

7. Repeat till convergence.

An important goal of this research is to simulate the rotational motion of the wind turbine rotor. To serve this purpose, a function in CFD codes called Multiple Reference Frames (MRF) is applied (it is named discontinuous mesh in SC/Tetra). Due to the rotation, two reference frames need to be established: one that is rotating with the wind turbine rotor and a second stationary reference frame relative to the ground. Note that the regular Navier-Stokes equations shown in Equations (2.1) and (2.2) can be applied to the stationary frame. However, to be applicable to the rotating reference frame, the momentum equations of the Navier-Stokes equations are modified as [20],

$$\rho \frac{D\vec{V}}{Dt} = -\nabla p + \mu \nabla^2 \vec{V} - \rho[\hat{\Omega} \times \hat{\Omega} \times \vec{r} + 2\hat{\Omega} \times \vec{V}], \quad (2.3)$$

where $\vec{r}$ is the position vector in the rotating frame, $\hat{\Omega} \times \hat{\Omega} \times \vec{r}$ is the centripetal force, and $2\hat{\Omega} \times \vec{V}$ is the Coriolis force caused by the rotational motion.
2.2 Geometric Model for S809 Airfoil

Model construction is the first step for computer simulations, especially for shape sensitive cases like wind turbine simulations. As mentioned in the previous chapter, there are two stages for this study, two-dimensional airfoil simulation and three-dimensional wind turbine simulation. Two-dimensional airfoil simulation is performed first because the lessons learn here can be passed on to the three-dimensional simulation. Also, errors in airfoil simulation can result in errors in turbine performance results.

The S809 airfoil is chosen for this research. This is a 21% thick airfoil designed by the National Renewable Energy Laboratory (NREL). It is one of the members of NREL’s thick-airfoil family, a group of airfoils specifically designed for horizontal axis wind turbines. The S809 airfoil is made to produce more lift force and lower profile drag force compared to the airfoils used for many aeronautical applications. A profile sketch of the S809 airfoil is shown in Figure 2.1.
SolidWorks, a parasolid based CAD software, is used to create the geometry of the S809 airfoil for importation to SC/Tetra or FLUENT. The SolidWorks model file is converted into an SC/Tetra compatible file via the converter software CADthru. By importing the airfoil profile coordinates provided in the NREL experiment report [11], shown in Appendix A, the S809 profile is generated. As shown in Figure 2.2, the two-dimensional model generally contains two parts, the inner disk with the airfoil profile and the outside domain. The 2m-diameter disk is used to execute different angles of attack from -1.04° to 20.16°, a total of 22 angles. The outside square domain must be made large enough so that the flow into the blades is steady and is not being affected by the walls of the domain. The entire model is given a z-axis (spanwise direction) thickness of 0.05m. Since the z-axis thickness is relatively small compared with the domain length in the x and y directions, the simulation is
considered pseudo two-dimensional. Also, the boundary conditions of the domain
walls are set up for a two-dimensional flow field that is discussed in a later section.

![Model construction in SolidWorks](image)

Figure 2.2   Model construction in SolidWorks.

SCTpre, the meshing tool of SC/Tetra, carries out the pre-processing jobs. The
next stage of model construction is mesh generation, which can profoundly affect the
results of the calculation. Three mesh formation types commonly used are structured
meshes, unstructured meshes, and hybrid meshes [22].

A structured mesh has regular grid elements arrayed in a uniform Cartesian
system. Grid elements are presented as quadrilaterals in two-dimensions and as
hexahedrons in three-dimensions. This form of connectivity has the advantages of
storage simplicity and suitability for the finite difference method. The value of
neighbor cells can be easily defined and accessed, which saves time and effort on calculations. The downside is that structured meshes are not suitable for solving complex geometries.

Opposite to structured meshes, elements of unstructured meshes are arranged irregularly and the element sizes can vary rapidly. Mesh elements are triangles in two-dimensions; tetrahedrons, pyramids or prisms in three-dimensions. Unstructured meshes conform to most geometries due to the flexibility of element shapes and sizes. However, since the mesh elements are not arranged in order, the connectivity between cells is stored explicitly; thus extra calculations are needed to determine the neighbor cells.

A hybrid mesh contains grid elements from the above two meshes. Edges from both methods are combined to perform a better mesh composition. For example, an unstructured mesh is applied in regions close to the airfoil because of its ability to capture complicated geometries. Due to the easy accessibility to the next cell value, structured meshes can be used in the outlying region of the computational domain since there is no complex shape to match.

Three different sizes of the outside square boundary are used to compare result independence to the far field boundary conditions. The computational domain sizes
tested are 10m x 10m, 50m x 50m, and 200m x 200m as illustrated in Figures 2.3, 2.4, and 2.5. Also, three different grid sizes are used; they are 180,000, 400,000, and 650,000 elements, respectively. Note that only the 10m x 10m domain is constructed in SolidWorks, whereas the 50m x 50m and 200m x 200m boundary sizes are domain expansions using a built-in function of SCTpre, creating elements by sweeping faces [17]. This function creates prismatic or hexahedral elements at selected surfaces, which provides features of a hybrid mesh. Using a larger mesh size out towards the boundaries allows the simulation to concentrate computer resources on the flow around the airfoil instead of distributing too much computer resources to the remote flow field.

Figure 2.6 shows a closer look at the mesh around the leading edge of the airfoil. It can be clearly seen that the mesh generation in SC/Tetra uses an unstructured mesh in this region. Five layers of prism elements are inserted at the wall of the airfoil to calculate the boundary layer with finer grids.
Figure 2.3  10m x 10m domain mesh.

Figure 2.4  50m x 50m domain mesh.

Figure 2.5  200m x 200m domain mesh.
2.3 Geometric Model for NREL Phase VI Wind Turbine Rotor

The NREL phase VI wind turbine is a test model designed for unsteady aerodynamic experiments (UAE), as shown in Figure 2.7, which has a rated power of 19.8 kW. The wind turbine has two blades using the S809 profile. Twist angle and chord length vary along the length of the blade. The rotor is equipped with pitch control, cone angle control, yaw system, and can operate in both upwind and downwind conditions. The rotational speed of the rotor during operation is 72 rpm.
Figure 2.7  Upwind view of NREL phase VI wind turbine in wind tunnel test [2].

The model construction of the three-dimensional wind turbine is also done in SolidWorks. The S809 airfoil geometry data is used (see Appendix A) since this airfoil shape is installed on the wind turbine blades. The building of the blade geometry is according to the blade dimension table provided in the wind tunnel test report [5] (see Appendix B). One single blade is 5.029 m in length and separated into 28 radial sections. Each cross section has a different chord length and twist angle, the twist axis is at 30% of the chord line from the leading edge. Similarly, to simulate the rotation of the rotor, the blade is placed in a disk first, as in the two-dimensional simulation, as shown in Figure 2.8. The rotational domain has a diameter of 15 m (about 1.5 times that of the rotor diameter) and thickness of 2.4 m (more than 10 times
the thickest part of the blade) to make sure the mesh around the blades are well
developed. The outside domain is a cylinder 55m in diameter and 66m in length,
which is built large enough to prevent the radial flow at the boundary from interfering
with the calculation.

For the three-dimensional simulations, two CFD solvers are used for calculations,
thus two different meshing tools are adopted; one for each solver. SCTpre was
introduced above and thus only the meshing tool used with FLUENT is discussed
here. The FLUENT meshing tool is ICEM CFD, a meshing software which can
generate both structured and unstructured meshes automatically.

Meshes around the blade’s cross section generated by SCTpre and ICEM CFD
are shown in Figures 2.9 and 2.10, respectively [21]. An unstructured mesh is used
due to the complicated geometry of the blade. It can be seen that the arrangement of
the grid elements is slightly different; tetrahedron elements in ICEM CFD seem to
have more regular organization. The grid element size gradually decreases from the
outside boundary towards the blades to have accurate calculation of the flow field
around the blades. Same as the method in the two-dimensional simulation, prism
layers are used in SC/Tetra and FLUENT to resolve the complex boundary layer at the
blade surface.
Figure 2.8  Rotor blade modeling in SolidWorks.

Figure 2.9  Mesh around cross sectional airfoil generated by SCTpre.
Figure 2.11 shows the assembly of the rotational domain (inner disk) and the stationary outside domain, as well as the mesh around the segment of the blade.

Figure 2.12 shows the entire model setup with a tetrahedron mesh. For this simulation, an unstructured mesh is constructed by SCTpre and includes approximately 2 million nodes, about 1.4 million nodes in the rotational disk region and 0.6 million nodes in the outside domain. The mesh constructed by ICEM CFD for FLUENT consists of about 2 million nodes, 1.3 million nodes are distributed in the rotational disk and 0.7 million nodes in the outside domain. By visual inspection it has been determined that the resolution of the mesh in the rotational disk should be above 1 million nodes so that the blade’s shape is not distorted, which would lead to inaccurate results.

Figure 2.10  Mesh around cross sectional airfoil generated by ICEM CFD.
2.4 Simulation Setup

After the mesh is correctly constructed, the simulation is ready to be given boundary conditions and other parameters required by CFD solvers to execute the calculation. Functions of two previously introduced CFD software packages, SC/Tetra and FLUENT, are discussed in this section.

2.4.1 Register Surfaces and Volume and Assign Material Properties

For two-dimensional simulation, the surface registration is generally displayed as in Figure 2.13. Note that the interface of the disk and the outer domain must be registered separately for later boundary condition setup.

Figure 2.11 Three-dimensional wind turbine model setup and blade segment in ICEM CFD.
Figure 2.12  Unstructured mesh construction in SCTpre.

Figure 2.13  Face registration of two-dimensional airfoil model.
The model consists of two volumes, the inner disk (*rotate*) and outer cylinder (*out_domain*) in the three-dimensional simulation; which represent the rotating part and the stationary part. The side of the cylinder, far upstream end of cylinder and far downstream end of the cylinder are registered as *domain_wall*, *domain_in*, and *domain_out*, respectively. Similar to the two-dimensional simulation, the surfaces of the disk and cylinder must be registered separately. The two blades are registered *bl1* and *bl2* as shown in Figure 2.14.
The selected fluid is incompressible air at 20°C with a density of 1.206 kg/m$^3$ and viscosity of $1.83 \times 10^{-5}$ m$^2$/s.

### 2.4.2 Analysis Condition

The two-dimensional simulation is a steady state simulation using the same Reynolds number, $1 \times 10^6$, as used in the Delft University’s wind tunnel experiment. The wind velocity is set at 15.17 m/s at the *inlet* face. The boundary condition at the *outlet* face is set to be the static pressure. The airfoil surface is the no-slip boundary condition. The two surfaces at the interface of the disk and domain, *side* and *inside* faces, are coupled for multiple reference frames. Although MRF is applied, it is only used to change the angle of attack and thus no rotational speed is given to the inner
disk. To achieve the pseudo two-dimensional simulation, the wall in the x-direction is set to a free slip wall condition. The convergence criterion is set to $1 \times 10^{-5}$.

The k-ε turbulence model and the Shear Stress Transport k-ω turbulence model are used in the two-dimensional simulation [23]. The k-ε model is a two-equation turbulence model that is widely used for engineering problems. This two-equation model has two equations to explain different variables in a turbulent flow: the first equation is usually kinetic energy and the second equation can be dissipation or viscosity. The two equations are

$$\begin{align*}
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k V_i) &= \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_l}{\sigma_k}) \frac{\partial k}{\partial x_j}] + G_k + G_b - \rho \varepsilon - Y_M + S_k, \\
&= (2.4)
\end{align*}$$

and

$$\begin{align*}
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon V_i) &= \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_l}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_j}] + C_{1\varepsilon} \frac{\varepsilon}{k} \varepsilon + (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon (2.5)
\end{align*}$$

where $G_k$ is the generation of turbulence kinetic energy caused by the mean velocity gradients, $G_b$ is the generation of turbulence kinetic energy due to buoyancy, $Y_M$ is the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, $C_{1\varepsilon}$, $C_{3\varepsilon}$ and $C_{2\varepsilon}$ are constants, $\sigma_k$ and $\sigma_\varepsilon$ are Prandtl numbers for $k$ and $\varepsilon$, $S_k$ and $S_\varepsilon$ are user-defined source terms.
In the SST $k-\omega$ model, the eddy viscosity is taken into consideration. In a fully developed turbulent flow, the effect of eddy viscosity becomes bigger than the molecular viscosity. Since the eddy viscosity is directly calculated, the condition of a turbulent flow is described clearly. The two transport equations express the kinetic energy, $k$, and specific dissipation rate, $\omega$, as:

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k V_j) = \frac{\partial}{\partial x_j} (\Gamma_k \frac{\partial k}{\partial x_j}) + G_k - Y_k + S_k,$$ (2.6)

and

$$\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_j} (\rho \omega V_j) = \frac{\partial}{\partial x_j} (\Gamma_\omega \frac{\partial \omega}{\partial x_j}) + G_\omega - Y_\omega + D_\omega + S_\omega,$$ (2.7)

where $G_\omega$ is the generation of specific dissipation rate, $\Gamma_k$ and $\Gamma_\omega$ are the effective diffusivities, $Y_k$ and $Y_\omega$ are the dissipation of $k$ and $\omega$, and $D_\omega$ is the cross-diffusion term. The Prantl number used for the SST $k-\omega$ model in the simulations is 0.9.

The two turbulence models described above can be applied to simulations in both commercial CFD codes used in this work. More detailed explanations of the turbulence models, such as developments and advanced applications can be found in Bardina et al. [24] and Menter [25,26].

The three-dimensional simulation is a transient analysis that uses a time step of $1 \times 10^{-4}$ seconds. Six different inlet velocities are used at \textit{domain in:} 7 m/s, 10 m/s, 13 m/s, 15 m/s, 20 m/s, and 25 m/s. Five inlet degrees, 0°, 10°, 20°, 30°, and 60°, are
applied to each wind speed to simulate yaw angles. The static pressure is set at the outlet surface \textit{domain\_out}. The convergence criterion is set as $1 \times 10^{-4}$. The rotating volume is 72rpm about the y-axis as in the wind tunnel test; three pairs of surfaces around the volume (\textit{inlet} and \textit{out\_inlet}, \textit{outlet} and \textit{out\_outlet}, \textit{side} and \textit{out\_side}) are defined as Multiple Reference Frames (MRF).

Mesh partition is a built-in function allowed in both SC/Tetra and FLUENT, which increases the speed of calculation and uses the CPU resources efficiently by decomposing the mesh into several sections of smaller meshes then distributing jobs to different processors to perform them in parallel. In this research, calculations are distributed to 8 to 32 processors when using SC/Tetra; 4 processors are the maximum allowed in FLUENT as shown in Figure 2.15.

Simulations are performed on 16 core, 2.70 GHz, 48 GB RAM machines; as well as the 92 nodes Taylor cluster at Wright State University. In the Wright State Taylor cluster, each node has a single processor that has a speed of 2.4 GHz and RAM storage of 1GB.
2.4.3 Post Processing

The converged results are interpreted and plotted by the post-processing tool. Visualized results such as velocity vectors, pressure contours, and streamlines are plotted and viewed in this work.
Chapter 3

Two-Dimensional Airfoil Studies

The results of CFD simulations on the S809 airfoil are exhibited and discussed in this chapter. Two convergence parameters that influence the accuracy, grid size and the computational domain size, and the turbulence model are studied in these simulations. These studies are focused on the aerodynamic characteristics of the S809 airfoil. The lift coefficient, drag coefficient and pressure coefficient are plotted and analyzed. Additional results like the pressure distribution and the velocity field around the airfoil are also shown.

All the two-dimensional cases are carried out by the commercial CFD software SC/Tetra. The results are compared to the results of the wind tunnel test at Delft University [11] and previous numerical work [13]. In this work the numerical simulations of the two-dimensional airfoil are set to calculate until steady state is reached. However, at higher angles of attack, due to the complicated flow conditions,
some regions of the flow field never reach a steady state condition.

SC/Tetra is capable of recording the airfoil characteristics for every iteration. The lift force and the drag force are the forces acting on the pressure center (quarter chord) that are normal and parallel to the wind direction, respectively. The pitching moment is monitored as the moment exerted around the z-axis. In this chapter, the lift, the drag and the pitching moment are expressed as dimensionless coefficients using Equations (1.6), (1.7), and (1.8).

3.1 Grid Size

The grid independence test is usually the first test to be done in a CFD simulation, and this is the case here as well. The purpose of the grid independence survey is to ensure the solution does not depend on the grid. The grid is simply a computational artifact to obtain the physical solution; thus the physical solution should be independent of the grid. This common CFD technique starts with a coarse mesh with refinements gradually applied until changes in the results are small enough that it can be said the grid does not affect the solution. The general way of refining grid points is to exponentially grow the mesh in each direction. However, refining a three-dimensional mesh by a factor of 2 can result in 8 times the number of grid

55
points, which for this problem is excessively time and resource consuming. The approach used in this work is to use a much smaller grid refinement. The mesh refinement used here only emphasizes the airfoil surface and the neighbor grids. The far field grid size remains the same.

Three different grid sizes are tested in this study. Test cases use a 10m x 10m domain size and the k-ε turbulence model. Each size of mesh is generated with a $y^+$ value between 5 and 100; this $y^+$ range is suitable for the turbulence model coupled with the near wall function. Thus there is enough resolution of the surface mesh for solving turbulence with the k-ε model [27]. All the cases are taken to convergence for the steady state analysis. Figures 3.1, 3.2, and 3.3 show the results of the grid independence test. Experimental results are also shown in these figures.

The results in Figures 3.1, 3.2, and 3.3 show that good convergence is obtained with all three grid sizes at lower angles of attack. At higher angles of attack there are some variations in the lift coefficient and drag coefficient results as a function of grid number. They start to separate at about an 8° angle of attack. The three grid numbers are all very close for all angles of attack for the moment coefficient. The reason the lift and drag coefficients become more sensitive to the grid after an 8° angle of attack is that flow separation starts to occur. Because of these differences at the higher
angles of attack, the finest grid tested, 650k grid points, is used for this work.

Figure 3.1  Lift coefficient as a function of angle of attack (AoA) and grid size.

Figure 3.2  Drag coefficient as a function of angle of attack (AoA) and grid size.
### 3.2 Turbulence Model

The selection of a turbulence model is the objective of this section. Two turbulence models are studied, the $k-\varepsilon$ model and the SST $k-\omega$ model. This study uses a 10m x 10m computational domain and the finest mesh from the previous grid independence study. The mesh for the boundary layer for the $k-\varepsilon$ model has a $y^+$ value between 30 and 100. For the SST $k-\omega$ model $y^+$ is set to approximately 2 because this turbulence model attempts to simulate eddy viscosity; thus the distance from the wall to the first node needs to be in the viscous sub-layer ($y^+ < 5$).

Figure 3.3  Moment coefficient as a function of angle of attack (AoA) and grid size.
Figure 3.4   Lift coefficients for different turbulence models.

Figure 3.5   Drag coefficients for different turbulence models.
The above figures show the performances of two turbulence models on the airfoil aerodynamic characteristics. It is clear that using the SST $k-\omega$ model produces better agreement to experimental results than using the $k-\varepsilon$ model. Some advantages and disadvantages of both models were observed during this convergence study.

The $k-\varepsilon$ turbulence model is robust, economical, and fast in terms of computational time due to its simplicity in the boundary layer. All the cases with the $k-\varepsilon$ model were able to reach small residuals with less calculation time than the SST $k-\omega$ model. However, the model reportedly has poor performance for solving complex flows like flow separation, strong streamline curvature, and strong pressure gradients.
It can be seen in Figures 3.4 and 3.5 that the k-ε model does not respond well to flow separation, but continues with a smooth trend [29].

The SST k-ω model seems to react to flow separation. This increased sensitivity of the SST k-ω model is due to its higher accuracy in the boundary layer. Because of the complexity at the boundary layer, the SST k-ω model is likely to over-predict eddies in this region and consumes more computational time to reach convergence [6]. After separation (angle of attack > 8°) a steady state analysis is no longer suitable for solving the flow field, thus a transient analysis is applied which results in some oscillations in the graphs.

3.3 Computational Domain Size

Another parameter that can possibly influence the accuracy and validity of the simulations is the size of the computational domain, which is easy to overlook among the many factors that influence a numerical simulation such as this. For a simulation with vortex effects like an airfoil, a proper outflow boundary should allow the flow to leave the computational domain with a discharge of vortices instead of interfering with the flow around the airfoil [29]. To achieve high accuracy, the computational domain sometimes needs to be very large; however, the larger size also increases the
cost of computation. In this convergence study, the computational domain is extended by using grid spacing with structured meshes as shown in Figures 2.3, 2.4, and 2.5; thus the computational time penalty is not great.

This convergence study continues using the most effective analysis conditions from the previous studies, the finest mesh and the SST k-ω turbulence model. Figures 3.7, 3.8, and 3.9 show that the numerical results oscillate at large angles of attack because this is an unsteady flow. As with the grid number study all cases tested compare well before separation takes place. When separation takes place there are differences in the results; but the results of a 50m x 50m domain size are similar to the 200m x 200m domain size results. This means a 50m x 50m domain is sufficient for these computations. However, to get the best accuracy, the results of a 200m x 200m domain size are used for comparison in later tests.

3.4 Comparisons to Other Investigations

According to the numerical results obtained from the previous convergence studies, the most accurate simulation setup is found using the SST k-ω turbulence model, 200m x 200m computational domain and 650,000 grid elements (with the
expansion of the domain, the grid size increases to 800,000 elements). In this section, the calculated results of this research are compared with experimental results and other computational simulations done by Wolfe and Ochs [12] and Hartwanger and Hovart [15]. Wolfe and Ochs [12] used CFD-ACE with a 10m x 10m computational domain, the k-ε turbulence model, and a Reynolds number of $2 \times 10^6$. The simulation of Hartwanger and Hovart [15] uses a 10m x 10m computational domain, 75,600 nodes, a Reynolds number of $1 \times 10^6$ and the SST k-ω turbulence model in the CFD software ANSYS-CFX.

![SST k-omega, 650,000 elements](figure)

**Figure 3.7** Lift coefficients for different domain sizes.
Figure 3.8 Drag coefficients for different domain sizes.

Figure 3.9 Moment coefficients for different domain sizes.
In Figure 3.10, all the numerical results have a good agreement with experimental results until an angle of attack of 6°, that is angles of attack where the flow is still attached to the surface. Flow separation is pronounced at 8° angle of attack but may begin at 6°. As discussed before, the k-ε model used in the simulation of Wolfe and Ochs [12] produce results that are larger than experimental results at higher angles of attack. Although Hartwanger and Hovart’s [15] SST k-ω model also over-predict the results, their curve is closer to experimental results than the results of Wolfe and Ochs. It can be said that Wolfe and Ochs low-resolution mesh converges easily, but has poor accuracy. The best comparisons to experimental work are produced with the simulation results done as part of this work.
For the drag coefficient, the trends are the same as for the lift coefficient. Once again the results obtained from the fully tested mesh of this research have the best comparison to experimental results.

Like the previous comparisons, the numerical results for the moment coefficient compare favorably to experimental results. This is true for all three simulation results. The only exception would be results for Hartwanger and Hovart [15] at angles of attack of $19^\circ$ and $20^\circ$, but even this deviation is not excessive.

Among all the coefficient comparisons, the k-$\varepsilon$ turbulence model shows the worst performance when compared to experimental results. As investigated in the
early convergence study and Wolfe and Ochs’ [12] conclusion, the k-ε turbulence model is not appropriate for solving separated flow. The simulation of Hartwanger and Hovart [15] shows that the SST k-ω turbulence model has good performance at small angles; however, discrepancies start to appear at large angles of attack. Furthermore, when it comes to the last few angles of attack, the results always have a sudden rise or drop. It is believed that the coarse mesh used by Hartwanger and Hovart [15] has difficulty simulating complex flows when the separation is fully developed. The simulations in this research seem to have the highest accuracy, which supports the analysis parameters chosen.

Figure 3.12  Comparison of moment coefficients.
Table 3.1  Percentage error between calculated aerodynamic coefficients.

<table>
<thead>
<tr>
<th>( \alpha )</th>
<th>( C_l )</th>
<th></th>
<th></th>
<th>( C_d )</th>
<th></th>
<th></th>
<th>( C_m )</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Sim.</td>
<td>Exp.</td>
<td>Error (%)</td>
<td>Sim.</td>
<td>Exp.</td>
<td>Error (%)</td>
<td>Sim.</td>
</tr>
<tr>
<td>-1.04</td>
<td>0.048</td>
<td>0.019</td>
<td>154.6</td>
<td>0.007</td>
<td>0.010</td>
<td>-21.4</td>
<td>-0.048</td>
</tr>
<tr>
<td>-0.01</td>
<td>0.165</td>
<td>0.139</td>
<td>18.9</td>
<td>0.009</td>
<td>0.009</td>
<td>-5.3</td>
<td>-0.051</td>
</tr>
<tr>
<td>1.02</td>
<td>0.266</td>
<td>0.258</td>
<td>3.0</td>
<td>0.009</td>
<td>0.010</td>
<td>-11.0</td>
<td>-0.054</td>
</tr>
<tr>
<td>2.05</td>
<td>0.406</td>
<td>0.378</td>
<td>7.4</td>
<td>0.009</td>
<td>0.010</td>
<td>-10.7</td>
<td>-0.057</td>
</tr>
<tr>
<td>3.07</td>
<td>0.497</td>
<td>0.497</td>
<td>0.0</td>
<td>0.010</td>
<td>0.010</td>
<td>-3.6</td>
<td>-0.060</td>
</tr>
<tr>
<td>4.1</td>
<td>0.618</td>
<td>0.617</td>
<td>0.2</td>
<td>0.011</td>
<td>0.010</td>
<td>6.6</td>
<td>-0.062</td>
</tr>
<tr>
<td>5.13</td>
<td>0.727</td>
<td>0.736</td>
<td>-1.2</td>
<td>0.017</td>
<td>0.010</td>
<td>70.4</td>
<td>-0.059</td>
</tr>
<tr>
<td>6.16</td>
<td>0.858</td>
<td>0.851</td>
<td>0.8</td>
<td>0.022</td>
<td>0.010</td>
<td>127.8</td>
<td>-0.057</td>
</tr>
<tr>
<td>7.18</td>
<td>0.915</td>
<td>0.913</td>
<td>0.2</td>
<td>0.024</td>
<td>0.013</td>
<td>91.1</td>
<td>-0.054</td>
</tr>
<tr>
<td>8.2</td>
<td>0.978</td>
<td>0.952</td>
<td>2.7</td>
<td>0.029</td>
<td>0.017</td>
<td>70.3</td>
<td>-0.046</td>
</tr>
<tr>
<td>9.21</td>
<td>1.090</td>
<td>0.973</td>
<td>12.0</td>
<td>0.034</td>
<td>0.025</td>
<td>39.1</td>
<td>-0.038</td>
</tr>
<tr>
<td>10.2</td>
<td>1.097</td>
<td>0.952</td>
<td>15.2</td>
<td>0.042</td>
<td>0.038</td>
<td>12.7</td>
<td>-0.037</td>
</tr>
<tr>
<td>11.21</td>
<td>1.057</td>
<td>0.947</td>
<td>11.7</td>
<td>0.055</td>
<td>0.073</td>
<td>-24.2</td>
<td>-0.036</td>
</tr>
<tr>
<td>12.23</td>
<td>1.014</td>
<td>1.007</td>
<td>0.7</td>
<td>0.070</td>
<td>0.064</td>
<td>9.6</td>
<td>-0.035</td>
</tr>
<tr>
<td>13.22</td>
<td>1.067</td>
<td>1.031</td>
<td>3.5</td>
<td>0.082</td>
<td>0.070</td>
<td>17.1</td>
<td>-0.040</td>
</tr>
<tr>
<td>14.23</td>
<td>1.032</td>
<td>1.055</td>
<td>-2.1</td>
<td>0.099</td>
<td>0.083</td>
<td>19.8</td>
<td>-0.042</td>
</tr>
<tr>
<td>15.23</td>
<td>0.985</td>
<td>1.062</td>
<td>-7.3</td>
<td>0.105</td>
<td>0.108</td>
<td>-3.1</td>
<td>-0.038</td>
</tr>
<tr>
<td>16.22</td>
<td>0.935</td>
<td>1.043</td>
<td>-10.4</td>
<td>0.138</td>
<td>0.143</td>
<td>-3.2</td>
<td>-0.035</td>
</tr>
<tr>
<td>17.21</td>
<td>0.951</td>
<td>0.969</td>
<td>-1.8</td>
<td>0.170</td>
<td>0.185</td>
<td>-8.5</td>
<td>-0.050</td>
</tr>
<tr>
<td>18.19</td>
<td>0.957</td>
<td>0.938</td>
<td>2.0</td>
<td>0.182</td>
<td>0.185</td>
<td>-1.8</td>
<td>-0.046</td>
</tr>
<tr>
<td>19.18</td>
<td>0.936</td>
<td>0.929</td>
<td>0.8</td>
<td>0.193</td>
<td>0.185</td>
<td>4.0</td>
<td>-0.057</td>
</tr>
<tr>
<td>20.16</td>
<td>0.996</td>
<td>0.923</td>
<td>8.0</td>
<td>0.181</td>
<td>0.185</td>
<td>-2.4</td>
<td>-0.069</td>
</tr>
</tbody>
</table>
Table 3.1 shows the percentage difference between numerical results from this work and Delft University wind tunnel test results. Except at a few angles, most the numerical results have an error within 20%.

### 3.5 Original SST k-ω Transition Turbulence Model

As mentioned above, even though the results show a good match to the experimental results with the advanced simulation setup, there are oscillations after an 8° angle of attack when using the SST k-ω model. It is expected that the flow separation starts to show up at around this angle of attack and thus the transition is the possible cause of the oscillations. The transition is the point where laminar flow turns into turbulence, which can profoundly affect the flow state and the fluctuations[30].

To improve the accuracy of these simulations, the user is able to alter the analysis somewhat using a Visual Basic interface. In this work the following three lines of code are added to the turbulence model function:

```python
SSTD
LORE 0
/
```

69
The purpose of adding this code was to change the corrected SST $k-\omega$ model in SC/Tetra back into original SST transition model to help the calculation pass through the transition point. Figures 3.13, 3.14, and 3.15 compare the calculated results using the original SST transition model results with experimental results and SC/Tetra SST $k-\omega$ model results.

It is clear that the corrected SST $k-\omega$ model has a better performance than the original version. The oscillations still exist and are even worse, especially for lift coefficients. For some reason, the original SST transition model has some similarities to the SST model of Hartwanger and Hovart [15]. Both simulations over-predict the lift coefficients, under-predict the drag coefficients and have similar trends on the moment coefficients. Moreover, it is noticeable that the values obtained with the original SST transition model also have sudden jumps at the last few angles. For this reason the original SST transition model is not used in this work.

3.6 Airfoil Characteristics

The pressure around the airfoil is expressed in the dimensionless form as shown in Equation (1.9). The pressure coefficient for the S809 airfoil is shown in Figures 3.16, 3.18, and 3.20. Velocity magnitude fields are shown in Figures 3.17, 3.19, and
Figure 3.13  Lift coefficient calculated with original SST transition model.

Figure 3.14  Drag coefficient calculated with original SST transition model
Figure 3.15  Moment coefficient calculated with original SST transition model.

Figure 3.16  Pressure distribution around S809 airfoil from -1.04° to 5.13°
3.21. Velocity magnitude contours allow the reader to observe how the flow condition changes with angle of attack.

The pressure distribution graphs are separated into three different angle groups, Figure 3.16 shows angles from -1.04° to 5.13°, Figure 3.18 shows angles of attacks from 6.16° to 12.23°, and Figure 3.20 shows angles of attack from 13.22° to 20.26°. The velocity magnitude plots show a velocity field for one angle of attack in each of these groups. All figures show the pressure being generally lower on the top of the airfoil as compared to the bottom of the airfoil. The largest pressure differences from the freestream value are at the leading edge and the differences taper off towards the...
trailing edge. The larger the angle of attack, the greater the pressure differences. From Figures 3.17 and 3.19 it can be seen that the low velocity profile at the trailing edge is moving towards the leading edge, this is a sign of the beginning of separated flow. It is also the point where the simulations become hard to converge and the calculated results start to oscillate. The larger the angle of attack, the earlier separated flow occurs on the back half of the upper surface as shown in Figure 3.20. Separated flow can be easily identified in Figure 3.21. The airfoil in Figure 3.21 is officially in the stalled condition.

Figure 3.18 Pressure distribution around S809 airfoil from 6.16° to 12.23°.
Figure 3.19  Velocity contour around S809 airfoil at 8.20°.

Figure 3.20  Pressure distribution around S809 airfoil for angles of attack from 13.22° to 20.16°.
Figure 3.21  Velocity contour around S809 airfoil at an angle of attack of 15.23°.
Chapter 4

Results of Three-Dimensional Wind Turbine Simulation

4.1 No Yaw Case

The objective of this three-dimensional simulation is to observe how the yaw angle affects a wind turbine’s power extracting ability. Some other graphical results are also presented. The simulations were performed for five yaw angles: 0°, 10°, 20°, 30°, and 60° yaw angle. In the sections below the 0° yaw angle is considered in the first section and the other yaw angles are considered in the second section. Results from the 0° yaw angle are considered separately because these results can be compared to experimental results and other computational results to check the validity of the CFD solvers used in this work. These checks will bear out the fact that there is a problem with SC/Tetra’s torque calculation.

After the performing the previous two-dimensional airfoil simulations, a great deal of experience was gained setting up the blade analysis conditions. This
experience was used to determine the three-dimensional simulation analysis conditions. The same grid size, boundary size, and turbulence model are used for the three-dimensional case as used for the two-dimensional case.

For the three-dimensional simulations, analyses were carried out with both SC/Tetra and FLUENT. A steady state solution was first obtained by SC/Tetra. A velocity inlet, static pressure outlet, natural inflow/outflow side domain values were applied as boundary conditions. The solver was able to reach convergence after approximately 1000 iterations. According to Equation (1.2), the power is equal to the shaft torque multiplied by the angular velocity. The angular velocity of the rotor in this simulation was set at 72rpm. Two of the primary global operating characteristics of the wind turbine blades that can easily be compared to experimental results are the shaft torque and the power extracted from the wind by the turbine blades.

The shaft torque is the moment that acts around the axis of rotation of the blades. The values of torque vary with the wind speeds shown in Figure 4.1. The shaft torques obtained by SC/Tetra were compared to results from experiments and the computational results of Hartwanger and Hovart [15]. The results show a good match between Hartwanger and Hovart’s [15] work and experiment; however, the comparisons between the calculated results from this work and the experimental
results are very poor. The torque magnitudes from this work for all but the lowest 7 m/s wind speed are very different than the experimental results and the computational results of Hartwanger and Hovart’s [15]. It is clear the shaft torque is supposed to increase from a wind speed of 7m/s to 10m/s; however, the trend of the numerical results from this work is decreasing.

![Figure 4.1](image)

**Figure 4.1** Shaft torques calculated by SC/Tetra.

To find the cause of the inaccuracy, many examinations and changes were applied to the SC/Tetra model. Methods from previous convergence studies were used; such as expanding the computational domain size three times, refining the mesh from
2 million to 5 million nodes, changing the analysis state, and changing the turbulence model. Although the values of torque increased a little with these techniques, the trend remained the same. The geometry, boundary conditions and solution setup were also double checked, but were determined to be correct. In Figure 4.2, the flow field around the rotor at 7 m/s shows that the rotation and boundary conditions were set up correctly.

Figure 4.2  Velocity field around the rotor at 7 m/s obtained from SC/Tetra.

To get a better handle on the inaccurate torque results produced by SC/Tetra a manual calculation of the shaft torque was done. This manual calculation of the torque utilized the blade pressures produced by SC/Tetra. Thus this hand calculation just
checks SC/Tetra’s ability to calculate the torque. To hand-calculate the moment that acts around the rotational axis of the rotor, a single blade was divided into 10 and 19 sections. The SC/Tetra produced pressures are roughly integrated over the sectional blade surface to find the forces that generate the torque around the rotational axis for the blade section. The moment at each section is the cross product of the force and the sectional moment arm relative to the rotational axis. The shaft torque is obtained by summing up the moments of all sections.

The hand-calculated torque is compared with experimental results and is shown in Figure 4.3. It can be seen that hand calculating the torque greatly improves the shaft torque values. Both the 10 and 19 section calculations have good agreement with experimental results at low wind speeds, but after 15m/s the hand calculated torques start to over-predict the experimental results. Never-the-less the hand calculated torques are a much better comparison to the experimentally determined toques than the values obtained directly from SC/Tetra. It must be understood that these hand calculations of torque are only approximate. These hand calculation results indicate that the torque calculation in SC/Tetra is wrong. It is unknown why the SC/Tet6ra torque calculation is wrong, but work done here indicates it is wrong. It is known that SC/Tetra calculates correct torques for other situations, which are simpler than the
wind turbine simulation being done here.

![Graph showing shaft torque vs wind speed](image)

**Figure 4.3** Torques acquired in a hand calculation using SC/Tetra produced pressure values.

To obtain accurate torque calculations, another CFD code, FLUENT, was adopted to continue the three-dimensional modeling of the two-bladed wind turbine. The same three-dimensional model geometry that was imported into SC/Tetra was imported into FLUENT. In addition, the same analysis and boundary conditions were used. ICEM CFD performed the meshing job for FLUENT, which generates a similar mesh to the one generated in SC/Tetra. The simulation was run as a transient analysis.

Figure 4.4 shows the shaft torques calculated by FLUENT. The numerical results
have good agreement with the experimental results at low wind speeds, especially at 13\text{m/s} to 15\text{m/s}. After 15\text{m/s}, over-prediction appears as the hand-calculation torques shown in Figure 4.3. It is believed the reason for these over predictions at high wind speeds is flow separation. According to the two-dimensional airfoil simulations presented in Chapter 3 of this thesis, results of the SST $k$-$\omega$ turbulence model tended to be unstable when flow separation occurred. In Figure 4.5, Equations (1.2) and (1.31) are applied to make the graph of power versus tip speed ratio. So the reader can see the precise differences between the simulated power coefficients and the experimental power coefficient, Table 4.1 is presented. From this table, the NREL Phase VI wind turbine’s highest efficiency is 36.96\% at a tip speed ratio of 5.5. A tip speed ratio corresponds to a wind speed of 7\text{m/s} at a wind turbine rotational speed of 72\text{rpm}. The FLUENT numerical result achieves a best efficiency of 26.86\% at 7\text{m/s}. The reader must recognize that while the power in the wind grows with the third power of the wind speed, the rotor’s ability to extract energy does not. This is especially true when separation occurs.

<table>
<thead>
<tr>
<th>Wind Speed (m/s)</th>
<th>7</th>
<th>10</th>
<th>13</th>
<th>15</th>
<th>20</th>
<th>25</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\eta$ (Experiment)</td>
<td>0.3696</td>
<td>0.2094</td>
<td>0.0940</td>
<td>0.0544</td>
<td>0.0219</td>
<td>0.0149</td>
</tr>
<tr>
<td>$\eta$ (Simulation)</td>
<td>0.2686</td>
<td>0.1458</td>
<td>0.0950</td>
<td>0.0526</td>
<td>0.0315</td>
<td>0.0226</td>
</tr>
</tbody>
</table>
Figure 4.4  Shaft torque calculated by FLUENT.

Figure 4.5  Power output of turbine as a function of the blade tip speed ratio.
To validate the accuracy of the simulation, pressure coefficients were calculated at 3 different sections of the blade, 30% R, 63% R, and 95% R. Note that in Equation (1.9) the pressure coefficients of a non-rotating blade are defined. When the airfoil is rotating, the rotational speed needs to be included in the pressure coefficient for the blade, thus Equation (1.9) becomes

\[ C_p = \frac{p - p_{\infty}}{\frac{1}{2} \rho (U + r\Omega)^2} \]  

(4.1)

The pressure coefficients determined numerically for the three-dimensional wind turbine blade calculations are shown in Figure 4.6. Pressure coefficients at 10m/s and 15m/s generally have a good match to experimental data. There are bigger discrepancies found for 10m/s at 0.95R and 15m/s at 0.3R, but overall the comparisons are quite good. This indicates the numerical results produced by FLUENT are good. Since the tip is the main region influencing the overall torque generation, the results of Figure 4.6 shed some light on the differences in experimental and simulated torque results shown in Figure 4.4. Because torque depends on distance from the rotational axis it is expected that pressure errors at smaller radial locations will cause less of an error in the torques than pressure errors at larger radial locations. This is why the difference in pressure for the 15m/s case at
0.3R is big, but the torque still matches the experimental result well. The discrepancy for 10m/s at 0.95R influences the torque more, which results in the larger error in torques shown in Figure 4.4. More pressure coefficient comparisons at different wind speeds and radial locations are also displayed.

Figure 4.7 shows the velocity field around the wind turbine at 7 m/s. From the upwind viewpoint of the rotor, it can be seen that rotor is indeed rotating which indicates the moving mesh routine works in FLUENT. Comparing Figure 4.9 with the velocity contours of SC/Tetra in Figures 4.2, the flow fields are similar and the velocity changes at the trailing edge are similar. Figure 4.9 also displays the side view of the computational domain. The wake after the rotor can be clearly observed; the wind velocity increases by the blade tips due to the rotation. It is evident that the computational domain is large enough for the downstream wake to be fully captured by the simulation.

The pressure contours on the blade surfaces are shown in Figures 4.10 to 4.15. It can be seen that the blade leading edge experiences the most pressure because of the rotation. With the increase of wind speed, the blades withstand more pressure along the leading edge. At the upper surface, unsteady pressure variations appear after 10m/s because of flow separation.
Figure 4.6  Pressure coefficients at 10m/s and 15 m/s.
Figure 4.7 Pressure coefficients at 7m/s and 13 m/s.
Figure 4.8  Pressure coefficients at 20 m/s and 25 m/s.
Figure 4.9  Flow field around the wind turbine at 7m/s. Upper figure is the top view and the lower figure is the side view.

Figure 4.10  Pressure contours in Pa at 7m/s (left is the lower surface of the blade and right is the upper surface of the blade).
Figure 4.11  Pressure contours in Pa at 10m/s (left is the lower surface of the blade and right is the upper surface of the blade).

Figure 4.12  Pressure contours in Pa at 13m/s (left is the lower surface of the blade and right is the upper surface of the blade).
Figure 4.13  Pressure contours in Pa at 15m/s (left is the lower surface of the blade and right is the upper surface of the blade).

Figure 4.14  Pressure contours in Pa at 20m/s (left is the lower surface of the blade and right is the upper surface of the blade).
4.2 Yaw Case Study

Now that the validity of the three-dimensional simulation has been established, a yaw angle study is performed. As discussed previously, a yaw system is used to keep the wind turbine rotor facing into the wind so that maximum power extraction from the wind is obtained. The yaw angle for a wind turbine from the top view is shown in Figure 4.16, yaw angles at downwind and upwind configuration are displayed. A yaw angle is the angle between the wind turbine centerline and the wind direction, the downwind configuration is used in this work. With misalignment to the wind, a wind turbine rotor can have a lower power extraction due to the yaw error. For the CFD simulation, changing the angle of a rotating mesh requires plenty of work. The
geometry of the rotating disk needs to be rebuilt and then remeshed. Therefore the easy way to simulate the yaw angle is to change the wind direction relative to the rotor. This is what is done in these simulations.

![Diagram of Turbine Yaw Angle](image)

Figure 4.16  Yaw angle.

For yaw angles between 0° and 30° similar trends in the torque are observed in Figure 4.15. When the yaw angle reaches 60°, the torque levels off as a function of wind speed. When the torque does not increase with wind speed, it can be said the blade has entered the post-stall state even at small wind speeds.

The power coefficients for different yaw angles as a function of wind speed are shown in Figure Comparison of power output with yaw angle. Like the torques the curves for yaw angles up to 30° have a similar shape. For a yaw angle of 60° the power as a function of wind speed is fairly constant. As expected, less power is obtained from the wind turbine as the yaw angle increases. Thus, as expected, it is
best to keep the wind turbine blades pointing directly into the wind. However, this plot does point out for these wind turbine blades that a slight misalignment of the turbine with the wind is not detrimental. Table 4. shows the numerical value of the power coefficients displayed in Figure 4.16 Comparison of power output with yaw angle. This table shows that the loss of power when the yaw angle is off by $20^\circ$, is less than 2%. For lower wind speeds the power loss is much less than this. In practice, considering the difficulty of measuring wind direction information and adjusting to this wind direction the yaw angle of a wind turbine seldom exceeds $10^\circ$.

![Figure 4.17](image)

**Figure 4.17** Calculated shaft torque changes with yaw angle.
Figure 4.18  Comparison of power output with yaw angle.

Table 4.2 Power coefficients for different yaw angles and wind speeds.

<table>
<thead>
<tr>
<th>Yaw Angle</th>
<th>Wind Speed</th>
<th>7m/s</th>
<th>10 m/s</th>
<th>13 m/s</th>
<th>15 m/s</th>
<th>20 m/s</th>
<th>25 m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>0.2686</td>
<td>0.1458</td>
<td>0.0950</td>
<td>0.0526</td>
<td>0.0315</td>
<td>0.0226</td>
<td></td>
</tr>
<tr>
<td>10°</td>
<td>0.2649</td>
<td>0.1385</td>
<td>0.0922</td>
<td>0.0467</td>
<td>0.0313</td>
<td>0.0176</td>
<td></td>
</tr>
<tr>
<td>20°</td>
<td>0.2584</td>
<td>0.1367</td>
<td>0.0909</td>
<td>0.0494</td>
<td>0.0300</td>
<td>0.0187</td>
<td></td>
</tr>
<tr>
<td>30°</td>
<td>0.2127</td>
<td>0.1184</td>
<td>0.0724</td>
<td>0.0396</td>
<td>0.0295</td>
<td>0.0176</td>
<td></td>
</tr>
<tr>
<td>60°</td>
<td>0.0871</td>
<td>0.1044</td>
<td>0.0434</td>
<td>0.0291</td>
<td>0.0088</td>
<td>0.0055</td>
<td></td>
</tr>
</tbody>
</table>
In this time of uncertain energy supplies it is important that renewable energy sources be investigated. This work is an attempt to advance renewable energies by developing a detailed computer model of the rotor of a wind turbine. This model simulates the aerodynamic characteristics of a wind turbine rotor. The results produced by this model, as well as the model itself are a small contribution to the renewable energy field.

Specifically this work presents a three-dimensional simulation of a two-bladed NREL wind turbine rotor that uses S809 airfoils. The model includes a detailed description of the flow field, pressure field, torques produced, and power produced for the entire rotor including the effects of blade rotation. Other parts of the wind turbine such as the nacelle, hub and tower are not included in this computer model. The only part of the wind turbine simulated is the rotor. The rotor is one of the most important parts of the wind turbine from the perspective of how do we extract the most possible
energy from the prevailing winds present at a given location. A number of computational results are presented for this wind turbine rotor and comparisons to experimental results and other simulation results are made. This thesis includes a detailed study of the effects of yaw angle on torque produced and power produced.

Before carrying out the three-dimensional simulation a two-dimensional simulation was performed. The two-dimensional simulation was developed to test the number of grid points required for an analysis such as this, the type of turbulence model that should be used to best predict the characteristics of the air flow through a wind turbine rotor, and the size of the computational domain required to perform a computation such as this. All of these studies should be useful to future wind turbine modelers. The two-dimensional study indicated that a mesh size of 650,000 grid elements is sufficient for calculations on this 10.058m diameter wind turbine rotor. Three different grid numbers were surveyed. The computation domain size chosen was 50m x 50m. This was more than large enough to ensure the far field boundary conditions had negligible effects on the flow results around the rotor blades. Three different domain sizes were studied. Lastly two different turbulent models were studied, the k-ε model and the SST k-ω model. The best turbulence model was chosen based on comparisons between calculated results and experimental results for the
S809 airfoil. The results indicated that both turbulence models do a good job for low angles of attack, but the SST \( k-\omega \) model performs much better for high angles of attack when flow separation becomes an issue. For this reason the SST \( k-\omega \) model was chosen for the three-dimensional simulations done in this work.

Before pressing on to the three-dimensional simulation, the two dimensional simulation results were compared to experimental results from Delft University and the National Renewable Energy Laboratory. Comparisons were made for the lift coefficient, the drag coefficient, and the moment coefficient. Very good comparisons were found for all three quantities with the best comparisons found for the moment coefficient. The comparisons for the lift and drag coefficient were outstanding for angles of attack below 8\(^\circ\), but started to show differences for angles of attack above this. These differences are more than likely caused by the difficulties of accurately modeling a separated flow over an airfoil. In general the simulation results were within 20% of the experimental results for most angles of attack. Only nine of the angles of attack simulated showed differences greater than 20%. These are considered good comparisons.

Experienced gained with the two-dimensional simulations was carried over to the three-dimensional simulations. In the three-dimensional simulations the added
difficulty of rotating blades was included in the model. This was handled by using two reference frames in the three-dimensional simulation, one stationary reference frame and one reference frame that rotated with the turbine blades. Accounting for the rotation of the blades greatly improves the physical fidelity of the model. It also increases the complexity of the computation.

For the three-dimensional simulation two CFD packages were used: SC/Tetra owned by the Cradle Software Corporation and FLUENT owned by the ANSYS Corporation. It was the original intent to only use SC/Tetra for the three-dimensional simulation, but some very unreasonable torque values were obtained from SC/Tetra. Some hand calculations of the torque using the pressure distributions on the blades produced by SC/Tetra leads the author to believe SC/Tetra has some problems in performing accurate torque values for the complex flow situation being studied in this work. It is known that SC/Tetra does produce accurate torque values for simpler flow cases, but it does not seem to be able to do it for the twisted wind turbine blade studies here. For this reason the CFD software FLUENT was adapted to perform the three-dimensional simulations required to complete this work. When compared to wind turbine experimental results, FLUENT is seen to produce very reasonable pressure distributions, torques, and powers. These comparisons helped to build
confidence in the use of the FLUENT model to perform the yaw study.

The yaw produced torque, power, and pressure coefficient information for yaw angles of $0^\circ$, $10^\circ$, $20^\circ$, $30^\circ$, and $60^\circ$ and wind speeds of $7m/s$, $10m/s$, $13m/s$, $15m/s$, $20m/s$, and $25m/s$. In general the higher the yaw angle the lower the torque and the lower the power extracted from the wind. This is what one would intuitively expect in regards to wind turbine operation. However, it cannot be intuitively deduced that for yaw angles up to $20^\circ$ the power extraction is only lowered by $1\%$ on average for the wind speeds simulated and by a maximum difference of $2\%$. This means that wind turbines do not have to be perfectly aligned with the wind to extract maximum powers. This is important because rapidly changing wind directions make it difficult to always align the turbine with the wind direction.

It is hoped that this research will be used to further develop CFD analysis of wind turbines. There are a number of investigators carrying out wind turbine simulations, but there needs to be more and the models need to become more sophisticated. It is the authors hope that more advances will be made in wind turbine simulation which will lead to improvements in wind turbine design. The wind turbine model in this research only has two blades. In future studies the model can be enhanced to include a hub, nacelle, tower, and additional blades. These wind turbine
parts do affect the performance of a wind turbine. Moreover, the effect of wind shear caused by the ground or influences between several wind turbines in a wind farm could be included.

More work needs to be done on turbulence models used for wind turbine simulation. In this work the SST $k-\omega$ model performed better than $k-\varepsilon$ model; however, the accuracy of SST $k-\omega$ model at large angle of attacks needs to be improved. Therefore, turbulence models that accurately track flow separation and transition is a good area for additional research.

Flow separation is not only a problem for computer simulation, but for wind turbine operation. Postponing separation will enhance power extraction. This could be done by a device installed on the blades or a new blade design. With the aid of computer simulation, new concepts can be easily tested.

To sum up, this work is an attempt to use CFD techniques to simulate a wind turbine and hopefully advanced the field of renewable energy. Due to some limitations, simulation in some ways is not accurate enough. Future work should focus on improving the accuracy of CFD calculations and predicting other phenomenon on a wind turbine. This work provides a good deal of information for future studies.
References


[10] Ingram, G., Wind Turbine Blade Analysis using the Blade Element Momentum Method. School of Engineering, Durham University, 2005


[12] Wolfe, W., Ochs, S., CFD Calculations of S809 Aerodynamic Characteristics,
AIAA Paper 97-0973, 1997


[23] Davidson, L., An Introduction to Turbulence Models, Department of Thermo and Fluid Dynamics, Chalmers University of Technology, 2011


[27] Salim, M., Cheah, S.C. Wall \( y^+ \) Strategy for Dealing with Wall-bounded Turbulent Flows IMECS, Hong Kong, 2009


APPENDIX A

S809 AIRFOIL GEOMETRY DATA FOR TWO-DIMENSIONAL SIMULATION MODELING

Table A.1 is the coordinates of the S809 airfoil designed and provided by NREL that is used for the 2D airfoil simulation modeling. The coordinates of upper surface and lower surface were imported separately into SolidWork to create curves then assembled as a full airfoil geometry.
### Table A.1 Airfoil coordinates.

<table>
<thead>
<tr>
<th>Upper Surface</th>
<th>Lower Surface</th>
</tr>
</thead>
<tbody>
<tr>
<td>x/c</td>
<td>y/c</td>
</tr>
<tr>
<td>0.00037</td>
<td>0.00275</td>
</tr>
<tr>
<td>0.00575</td>
<td>0.01166</td>
</tr>
<tr>
<td>0.01626</td>
<td>0.02133</td>
</tr>
<tr>
<td>0.03158</td>
<td>0.03136</td>
</tr>
<tr>
<td>0.05147</td>
<td>0.04143</td>
</tr>
<tr>
<td>0.07568</td>
<td>0.05132</td>
</tr>
<tr>
<td>0.1039</td>
<td>0.06082</td>
</tr>
<tr>
<td>0.1358</td>
<td>0.06972</td>
</tr>
<tr>
<td>0.17103</td>
<td>0.07786</td>
</tr>
<tr>
<td>0.2092</td>
<td>0.08505</td>
</tr>
<tr>
<td>0.24987</td>
<td>0.09113</td>
</tr>
<tr>
<td>0.29259</td>
<td>0.09594</td>
</tr>
<tr>
<td>0.33689</td>
<td>0.09933</td>
</tr>
<tr>
<td>0.38223</td>
<td>0.10109</td>
</tr>
<tr>
<td>0.42809</td>
<td>0.10101</td>
</tr>
<tr>
<td>0.47384</td>
<td>0.09843</td>
</tr>
<tr>
<td>0.52005</td>
<td>0.09237</td>
</tr>
<tr>
<td>0.56801</td>
<td>0.08356</td>
</tr>
<tr>
<td>0.61747</td>
<td>0.07379</td>
</tr>
<tr>
<td>0.66718</td>
<td>0.06403</td>
</tr>
<tr>
<td>0.71606</td>
<td>0.05462</td>
</tr>
<tr>
<td>0.76314</td>
<td>0.04578</td>
</tr>
<tr>
<td>0.80756</td>
<td>0.03761</td>
</tr>
<tr>
<td>0.84854</td>
<td>0.03017</td>
</tr>
<tr>
<td>0.88537</td>
<td>0.02335</td>
</tr>
<tr>
<td>0.91763</td>
<td>0.01694</td>
</tr>
<tr>
<td>0.94523</td>
<td>0.01101</td>
</tr>
<tr>
<td>0.96799</td>
<td>0.006</td>
</tr>
<tr>
<td>0.98528</td>
<td>0.00245</td>
</tr>
<tr>
<td>0.99623</td>
<td>0.00054</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
</tr>
</tbody>
</table>
APPENDIX B

SECTIONAL GEOMETRY DATA OF NREL PHASE VI WIND TURBINE BLADE
FOR THREE-DIMENSIONAL SIMULATION MODELING

To model NREL Phase VI wind turbine blades, the airfoil coordinates from Appendix A is acquired. The blade consists of airfoil with various chord length and twist at each radial section. The blade is attached to the hub at the blade radius of 0.508m. From 0.508m to 0.883m, there is a cylindrical section for the root of the blade. The shape of airfoil starts to develop after 0.883m. The twist angle is positive towards upwind direction.
Table B.1 Blade geometry data.

<table>
<thead>
<tr>
<th>Radial Distance r (m)</th>
<th>Span Station (r/5.029 m)</th>
<th>Chord Length (m)</th>
<th>Twist (degrees)</th>
<th>Thickness (m)</th>
<th>Twist Axis (% chord)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>Hub Center</td>
<td>Hub Center</td>
<td>Hub Center</td>
<td>Hub Center</td>
</tr>
<tr>
<td>0.508</td>
<td>0.101</td>
<td>0.218</td>
<td>0</td>
<td>0.218</td>
<td>50</td>
</tr>
<tr>
<td>0.66</td>
<td>0.131</td>
<td>0.218</td>
<td>0</td>
<td>0.218</td>
<td>50</td>
</tr>
<tr>
<td>0.883</td>
<td>0.176</td>
<td>0.183</td>
<td>0</td>
<td>0.183</td>
<td>50</td>
</tr>
<tr>
<td>1.008</td>
<td>0.2</td>
<td>0.349</td>
<td>6.7</td>
<td>0.163</td>
<td>35.9</td>
</tr>
<tr>
<td>1.067</td>
<td>0.212</td>
<td>0.441</td>
<td>9.9</td>
<td>0.154</td>
<td>33.5</td>
</tr>
<tr>
<td>1.133</td>
<td>0.225</td>
<td>0.544</td>
<td>13.4</td>
<td>0.154</td>
<td>31.9</td>
</tr>
<tr>
<td>1.257</td>
<td>0.25</td>
<td>0.737</td>
<td>20.04</td>
<td>0.154</td>
<td>30</td>
</tr>
<tr>
<td>1.343</td>
<td>0.267</td>
<td>0.728</td>
<td>18.074</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>1.51</td>
<td>0.3</td>
<td>0.711</td>
<td>14.292</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>1.648</td>
<td>0.328</td>
<td>0.697</td>
<td>11.909</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>1.952</td>
<td>0.388</td>
<td>0.666</td>
<td>7.979</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>2.257</td>
<td>0.449</td>
<td>0.636</td>
<td>5.308</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>2.343</td>
<td>0.466</td>
<td>0.627</td>
<td>4.715</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>2.562</td>
<td>0.509</td>
<td>0.605</td>
<td>3.425</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>2.867</td>
<td>0.57</td>
<td>0.574</td>
<td>2.083</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>3.172</td>
<td>0.631</td>
<td>0.543</td>
<td>1.15</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>3.185</td>
<td>0.633</td>
<td>0.542</td>
<td>1.115</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>3.476</td>
<td>0.691</td>
<td>0.512</td>
<td>0.494</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>3.781</td>
<td>0.752</td>
<td>0.482</td>
<td>-0.015</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>4.023</td>
<td>0.8</td>
<td>0.457</td>
<td>-0.381</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>4.086</td>
<td>0.812</td>
<td>0.451</td>
<td>-0.475</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>4.391</td>
<td>0.873</td>
<td>0.42</td>
<td>-0.92</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>4.696</td>
<td>0.934</td>
<td>0.389</td>
<td>-1.352</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>4.78</td>
<td>0.95</td>
<td>0.381</td>
<td>-1.469</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>5</td>
<td>0.994</td>
<td>0.358</td>
<td>-1.775</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
<tr>
<td>5.029</td>
<td>1</td>
<td>0.355</td>
<td>-1.815</td>
<td>20.95% chord</td>
<td>30</td>
</tr>
</tbody>
</table>