k-ε turbulence modeling for a wind turbine

Comparison of RANS simulations with ECN wind turbine test site Wieringermeer (EWTW) measurements

ERMAN EREK

Master of Science Thesis
EGI-2011-020MSC EKV833
Stockholm, Sweden 2011
$k$-$\varepsilon$ turbulence modeling for a wind turbine:
Comparison of RANS simulations with ECN wind
turbine test site Wieringermeer (EWTW)
measurements

Erman EREK
Master of Science Thesis

Supervisors:
Dr. Arno Brand (ECN)
Dr. Damian Vogt (KTH)

KTH School of Industrial Engineering and Management
Energy Technology EGI-2011-020MSC EKV833
Division of Heat and Power Technology
SE-100 44 STOCKHOLM
2011
Abstract

This report is the final requirement for completion of MSc Degree in Sustainable Energy Engineering at KTH, Royal Institute of Technology, Stockholm. It was accomplished at Energy Research Center of the Netherlands (ECN) under supervision of Dr. Arno Brand at ECN and Dr. Damian Vogt at KTH within a research project on modeling of turbulence near a wind turbine. In this report we discuss the use of $k$-$\varepsilon$ RANS (Reynolds-averaged Navier-Stokes equations) turbulence model for wind turbine applications. This model has been implemented in the new wind turbine wake CFD code that is being developed at ECN. Simulations of the wind turbine test site EWTW are compared with measurements conducted between 2005 and 2009 and with FarmFlow, ECN’s current wind turbine wake code. Based on the results the uncertainties in the current approach are highlighted and areas for possible improvement are discussed.
## Contents

1 Introduction .................................................. 4
   1.1 Background ............................................. 4
   1.2 Physics of the problem .................................. 6
   1.3 Existing models ........................................ 12
      1.3.1 CFD ................................................. 13
   1.4 Current approach ....................................... 15

2 Discretization of RANS equations .......................... 16
   2.1 Navier-Stokes equations ................................ 16
   2.2 $k$-$\varepsilon$ model .................................. 17
   2.3 Boundary conditions .................................... 18
   2.4 Actuator disc model .................................... 20

3 Code verification ............................................. 21
   3.1 Lid-driven cavity ....................................... 21
   3.2 Actuator disc ........................................... 25

4 Results ......................................................... 29
   4.1 ECN EWTW measurements ................................. 29
   4.2 RANS simulations ........................................ 34

5 Conclusions .................................................. 47
1 Introduction

1.1 Background

Wind energy has been traditionally used as a source for usable energy, e.g. to propel boats with the help of sails, to grind grain with the help of wind mills, or to pump water as in the case of historically widely-applied water drainage applications of the Dutch using wind mills. The earliest examples of wind mills were of vertical axis type. The first horizontal axis wind mills appeared in 1000’s AD, of which some arrived in Europe around 1150-1200. Those are examples of usable mechanical energy that was converted from the kinetic energy of the wind. Later on, another way of utilizing this kinetic energy was found: conversion to electrical energy using wind turbines. The first wind turbines generating electricity date back to the late 1800’s. For example, wind turbines with DC generators supplied some 3% of the electricity demand in Denmark in 1918. Typical sizes of them were between 20 and 35kW with a total capacity of around 3MW in Denmark. Later on, following the continuous development in the field, the largest wind turbine size has reached 7MW today. As of the end of 2007, the total installed capacity world wide reached 94GW whereas the average growth rate of installed capacity was 24% over the period of 2002-2007 [Ackermann, 2008]. In 2008, the total installed capacity world wide reached 121GW in 2008. In the same year the Netherlands reached an all-time record of 490MW of new installed wind power capacity. Among International Energy Agency (IEA) member countries, the total installed capacity has increased from less than 5GW in 1995 to nearly 92GW in 2008 [IEA, 2008a]. According to IEA’s projections for 2030 based on its Reference Scenario, the world’s wind energy output will increase eleven-fold compared with the 2006 output [IEA, 2008b].

The clues given here about the situation and future of wind energy show clearly that the research in the field ongoing and in the future has an increasing importance. As more wind energy projects are on the way, investors, project developers and scientists desire
to have a deeper understanding of the nature of wind and related technology. It is of their interest to have the scientific knowledge, utilities and tools for acquiring best economically feasible solutions to design and manufacture turbines, predict wind energy output for certain time periods, position wind turbines in certain ways in wind farms for optimum output and decrease the costs involved.

While wind turbines extract energy from the wind they reduce the kinetic energy that is left downstream of the turbine. Wind turbine wake modeling is necessary for estimating the kinetic energy left for downstream turbines. This is important because wake losses in a wind farm can reach up to 40%, whereas it is 8% and 12% on average over different wind directions for onshore and offshore farms, respectively [Sanderse, 2009, Barthelmie, 2008, Barthelmie, 2009]. In addition, the grouping of wind turbines may lead to increased dynamic loads on the blades due to higher turbulence levels [Sanderse, 2009]. Thus, wake effects play a big role in optimizing the wind farm layout. Therefore, developers give continuous effort on improving wake models with less computation costs and more detailed and accurate results, which as a result may help them in increasing power output and decreasing the maintenance and operational costs by reducing the fatigue loads. Additionally, Annual Energy Production (AEP) of a wind farm must be as accurate as possible and it is a necessity for obtaining finance from banks for a given project [Réthoré, 2009]. Given these facts, wind farm wake modeling is an important tool to estimate the power generation capacity of a wind farm and the loading on the turbines with a given wind inflow.

ECN, Energy Research Center of the Netherlands, located in Petten in Noord-Holland Province of the Netherlands, is the leading research institute of the country in the fields of sustainable energy, energy conservation and clean use of fossil fuels. ECN’s mission is to develop high-level knowledge and technology for a sustainable energy system, and transfer them to the market. ECN has the following units: Policy Studies, Hydrogen & Clean Fossil Fuels, Efficiency & Infrastructure, Engineering & Services, Biomass, Coal
& Environmental Research, Solar Energy, and Wind Energy.

Wind Energy Unit of ECN conducts research in wind energy that can be said to have entered the phase of technological maturity. The priority of the wind sector is to install and connect large amounts of wind power to the grid quickly, safely and cost effectively, whilst simultaneously addressing concerns about the turbine reliability, availability and accessibility.

This thesis starts with an introduction of the physics of the problem, existing wake and turbulence models, and continues with the current approach. Later, implementation of \( k-\varepsilon \) turbulence model to the Computational Fluid Dynamics (CFD) code [Sanderse, tbp] that is in development stage at Wind Energy Unit of Energy Research Center of the Netherlands (ECN) is explained. This code utilizes Finite Volume Method (FVM) of second order for incompressible Navier-Stokes equations with staggered, uniform/non-uniform grids in 2D and general boundary conditions and body force. The laminar flow performance of the code is verified by assessing the results for two different laminar flow cases with benchmark results from literature. Finally, results obtained from the turbulent wake model by solving the Reynolds-averaged Navier-Stokes equations (RANS) with the \( k-\varepsilon \) model will be compared with ECN Test Site Wieringermeer (EWTW) measurements and the FarmFlow code of the ECN Wind Energy Unit, and conclusions are drawn.

1.2 Physics of the problem

To extract energy from wind, a wind turbine reduces the wind speed significantly and causes modifications in the structure of the flow due to small turbulence structures. At the boundaries of the wake region vortex structures occur due to the velocity difference between the wake region and free-stream region. Turbulent vortex structures increase in size throughout the wake interface expansion and help the wake to recover. Interaction of atmospheric turbulence and these various turbulence structures leads to a
spatial oscillation of the wake. This phenomenon is called wake meandering and con-
sidered as a large-scale component of the wake turbulence [Réthoré, 2009]. These
modifications applied on the flow by a wind turbine make any downstream wind turbine
get an inflow with a different mean wind velocity and turbulence intensity. Therefore,
structural vibrations are observed (with wind fluctuations), as well as increased fatigue
loads. The atmospheric turbulence scale is for example of the order of several hundred
meters whereas wake interface turbulence scale, which is characterizing the mixing of
the wake boundaries with the atmospheric flow, is roughly of the order of the size of the
rotor diameter. Blade induced vortex structures however are of the order of the dimen-
sion of blade chord. To simplify the complexity involved in these different length scales,
close and far wake regions are defined as the direct vicinity of the turbine and regions
further downstream of the turbine, respectively. In the close wake region blade induced
turbulence structures are dominant and in the far wake region the shear induced turbu-
lence is dominant whereas it is progressively absorbed by the atmospheric turbulence
[Réthoré, 2009].
Wind turbines are operating in the lowest part of the atmospheric boundary layer (ABL),
which is often modeled as:

$$\bar{u}(z) = u^* (\ln(z/z_0) + \Psi)/\kappa,$$  \hspace{1cm} (1)

$z_0$ being the surface roughness length that is in a wide range from 0.001m for rough
sea, 0.03m for areas with low vegetation to 1-2m for urban areas with low-rise buildings
or large forests with clearings, $u^*$ being the friction velocity, $\kappa$ the von Kármán constant
and $\Psi$ a function depending on atmospheric stability. $u^*$ is defined as $u^* = \sqrt{\tau_w/\rho}$,
where $\tau_w$ is the wall shear stress.
The height of the atmospheric boundary layer has a range from hundred meters up to
several hundred meters or few kilometers. The turbulence intensity and the height of
the ABL are affected by the thermal stratification. The atmospheric boundary layer can
be neutral, stable, or unstable depending on whether the adiabatic lapse rate defined as the change in temperature with height for a system without heat exchange, \( \left( \frac{dT}{dz} \right)_{ad} \) \((-1\, ^\circ C\, per\, 100m\, approximately)\) is equal to or is larger or smaller than the actual lapse rate, \( \left( \frac{dT}{dz} \right) \) \((-0.65\, ^\circ C\, per\, 100m\, on\, average)\).

The stability of the atmosphere is indicated with the (gradient) Richardson number:

\[
R_i = \frac{g}{T_0 \left( \frac{dT}{dz} \right)^2}.
\]  

(2)

where \( g \) is the gravitational acceleration, \( T_0 \) a reference temperature, \( T \) the mean potential temperature and \( z \) the height above the ground. It indicates the ratio between thermally created turbulence because of buoyancy and mechanically created turbulence because of velocity shear. Theoretically it equals zero at neutral stratification.

Atmospheric turbulence is a function of surface roughness length, atmospheric stability, distance above the ground and is anisotropic. Turbulence intensity is defined as:

\[
I = \frac{\sigma}{\bar{u}}.
\]  

(3)

where

\( \sigma \): the standard deviation of the wind velocity in the average wind direction
\( \bar{u} \): the magnitude of the average wind velocity

Turbulence intensity decreases with height within the ABL.

Considering a flow passing through a wind turbine rotor, the very first observation is that a fluid element loses some of its kinetic energy while it passes through the rotor of a wind turbine. Consequently the flow slows down gradually from its upstream value \( u_\infty \) to an average downstream value \( u_w \) in the far wake. According to the actuator disc theory, taking a wind turbine as black-box and considering the flow in one dimension only the following analytical equations result from conservation of mass, momentum
and energy for incompressible flows:

\[ \dot{m} = \rho A_{\infty} u_{\infty} = \rho A_d u_d = \rho A_w u_w \]  
\text{(4)}

\textbf{Continuity:}

\[ T = \dot{m}(u_{\infty} - u_w) = (p_d^+ - p_d^-)A_d \]  
\text{(5)}

\textbf{Force on disc:}

\[ E = \frac{1}{2} m(u_{\infty}^2 - u_w^2) \]  
\text{(6)}

\textbf{Energy extracted:}

![Diagram of 1D flow through an actuator disc [Sanderse, 2009]]

The power from this velocity reduction and the mass flow is:

\[ P = \frac{1}{2} \dot{m}(u_{\infty}^2 - u_w^2), \]

which equals the power performed by the force \( T \) acting on the disk, \( P = Tu_d = \dot{m}(u_{\infty} - u_w)u_d \). Thus, \( u_d = \frac{1}{2}(u_{\infty} + u_w) \).

The \textit{power} coefficient \( C_P \) is defined as the non-dimensionalized power by the wind speed and the rotor swept area:

\[ C_P = \frac{P}{\frac{1}{2} \rho u_{\infty}^3 A_d} = \frac{P}{\frac{1}{2} \rho u_{\infty}^3 A_d} \Rightarrow C_P = (u_{\infty} + u_w)(u_{\infty}^2 - u_w^2)/u_{\infty}^3 = \frac{1}{2}(1 + b)(1 - b^2) = 4a(1 - a)^2 \]

where \( a = 1 - \frac{u_d}{u_{\infty}} \) is the axial induction factor and \( b = \frac{u_w}{u_{\infty}} \). Thus, \( a = (1 - b)/2 \). The \textit{Betz} limit is defined as \( C_{P_{\text{max}}} = \frac{16}{27} = 0.59 \rightarrow 59\% \) at \( a = b = \frac{1}{3} \) where a maximum of 59% of the wind energy can be converted into mechanical power. Similarly, the \textit{thrust} coefficient is defined as \( C_T = \frac{T}{\frac{1}{2} \rho u_{\infty}^2 A_d} = 1 - b^2 = 4a(1 - a) \).

Due to the velocity reduction behind the wind turbine, i.e. \( u_w < u_d \Rightarrow A_w > A_d \), the
wake expands. The wake expansion is larger for larger $C_T$.

At the actuator disc location the pressure drops suddenly from $p_d^+$ to $p_d^-$, and it gradually recovers from $p_d^-$ to $p_\infty$ further downstream in the wake as seen in the Figure 1. The curvature of the streamlines support the pressure difference between the near wake and the free-stream region.

Taking an inviscid vortex system into account the velocity develops in the wake as:

$$u_i/u_\infty = 1 - \frac{1 - b}{2} \left(1 + \frac{2x}{\sqrt{1 + 4x^2}}\right),$$

(7)

where:

- $u_i$: velocity in the wake
- $x$: the non-dimensionalized distance from the rotor
- $b = \sqrt{1 - C_T}$.  

![Figure 2: Wake velocity development for various $C_T$ values [Sanderse, 2009]](image)

At two diameters downstream of the rotor, the velocity reaches almost $u_w$ as shown in the Figure 2. $u_w$ can be used in the calculation of the power output of the next actuator disc in the wake.
In the actuator disc model there is no tangential velocity component in the wake as the shape of the energy converter is not taken into account. In reality, the rotor has a number of blades which apply a reaction torque on the flow so that it rotates in the direction opposite to the rotor. The tangential velocity concerning this rotation is not taken into account in actuator disc model.

Although there is not a definite distinction between near wake and far wake of a wind turbine, the near wake is typically defined as the region up to $2D$ distance downstream of the rotor, whereas far wake is the region further downstream. In the near wake the shape of the flow field is determined by the turbine geometry so that the performance of the turbine is determined. In the far wake the rotor shape has less effect on the shape of the flow field whereas the turbulence and wake modeling are still of importance.

The velocity difference between the free-stream region and the wake causes a shear layer where turbulent eddies are formed. This layer becomes thicker downstream. Ambient shear flow interacts with the shear layer so that it becomes non-uniform. The turbulence intensity in the upper part is larger than the lower part so that there exist two peaks in the turbulence intensity in the wake whereas these peaks can no longer be distinguished in the far wake. For larger ambient turbulence levels the maximum velocity deficit is reached at 1-2 rotor diameter downstream of the turbine which is in the near wake. This length may be longer for low ambient turbulence levels [Ainslie, 1988]. $2.25D$ is considered to be the distance that the wake is fully expanded based on experiments [Schepers, 2003].

The thrust on the rotor determines the wake velocity $u_w$ and the shear between wake and free-stream. Larger thrust results in lower wake velocity and larger shear. At high levels of rotor loading, the part of the kinetic energy of the flow that is converted into large-scale turbulent motion is notably large in amount. This leads to the turbulent wake state where the axial induction factor $\alpha > 0.5$, typically. The efficient mixing of the low velocity fluid of the wake and the high velocity fluid of the free-stream region by the
turbulence results in the momentum transfer into the wake so that the wake expands and the velocity deficit is reduced.

1.3 Existing models

Most of the wake models are based on the principles of conservation of mass and momentum, especially CFD models, although widely used engineering models differ from the rest in the sense that they are not directly based on such principles. Turbines are modeled as momentum sinks similar to the thrust force of the turbine. Some terms in the Navier-Stokes equations are neglected in simplest models assuming the wake homogeneous and axisymmetric [Réthoré, 2009].

Fastest wake models in terms of computation time utilize methods to model wake expansion. These methods apply conservation of momentum on a wake region that is increased according to a semi-empirical wake expansion function [Réthoré, 2009]. Simplified turbulence closure techniques such as eddy-viscosity mixing-length models have been also used to attack the problem of turbulence. When there is small scale velocity changes eddy-viscosity models have difficulties to model large-scale turbulence. It is possible to split the turbulence in to two, one for large scale part and one for small scale part modeled with an eddy-viscosity model [Réthoré, 2009].

There are numerous wake models that contain all components of the Navier-Stokes equations. They make use of different ways to model atmospheric turbulence and the wake. For instance, CFD with Large Eddy Simulation (LES), Reynolds-averaged Navier-Stokes (RANS), and Reynolds-stresses Model (RSM) with different solvers and force estimation algorithm. Complexity of the model and accuracy of the results increase the computational costs as expected [Réthoré, 2009].
1.3.1 CFD

Making use of *Computational Fluid Dynamics*, solutions of any accuracy can be attained in principle when the governing equations are known accurately as in the case of Navier-Stokes equations for incompressible Newtonian fluids. For phenomena like turbulence, the direct numerical simulation for large wind turbines is not feasible though. This brings the essential introduction of turbulence models, which give approximate solutions as a representation of the reality. In solutions obtained with models there are discretization errors and errors involved in solving the discretized equations, so that the validation of the models with experiments is an inevitable necessity. Discretization errors occur due to the fact that CFD methods solve the Navier-Stokes equations using approximations based on a system of algebraic equations that are applied to small domains in space and time.

The general conservation equations for mass and momentum can be simplified for incompressibility of some fluids such as liquids as well as gases for certain flows. If the *Mach* number is below 0.3 the flows are said to be incompressible, which is the case for flows around wind turbines in the standard atmosphere at sea level although some compressibility effects may occur at blade tips if the rotor should be modeled directly. As the velocities in the wake are quite lower than the limit of 100 m/s for *Mach* = 0.3 the incompressibility assumption is valid for wind farm wakes.

The *Reynolds number* is the dimensionless parameter of the incompressible Navier-Stokes equations, which turns out to be of the order $O(10^6)$ based on the chord and rotational speed at a certain radius in large wind turbine applications. Together with the fact that the inflow conditions are turbulent due to the atmospheric conditions, and the adverse pressure gradient, this high Reynolds number results in turbulent flow over a wind turbine blade. *Energy dissipation* takes places at very small scales in turbulent flows, e.g. the smallest eddy size: $\eta \sim Re^{-3/4}$, when compared with laminar flows, which results in very complex and in-practice impossible *Direct Numerical Simulation*. 

13
The behavior of such small scale eddies can only be modeled by the closure models such as Large Eddy Simulation (LES), or Reynolds-averaged Navier-Stokes (RANS) as mentioned earlier.

RANS models rely on averaging concepts introduced by Reynolds (1895), which in general can be described by the term “mean”. *Time averaging* is one of several averaging processes that Reynolds introduced, and is appropriate in stationary turbulence applications. In principle the variables are split in an average and a fluctuation. The fundamental problem of turbulence lies in the need of a prescription for computing the *Reynolds-stress tensor* that is shown in section “Discretization of RANS equations”, so that all mean properties of the turbulent flow under consideration can be computed. This Reynolds-averaging process produces six unknown parameters from the Reynolds-stress tensor and four others from mean-flow properties, i.e. pressure and three velocity components in 3-D, for which there are not enough number of equations [Wilcox, 1993]. Thus, this is the so called *closure problem* of turbulence that requires appropriate determination of additional equations and corresponding parameters. Zero-, one-, two-equation models refer to the number of partial differential equations needed to complete a RANS model, most of which rely on the *Boussinesq hypothesis* that states that small-scale turbulent should be linearly proportional to large ones [McDonough, 2004]. In fact there is no physical reason for this hypothesis to hold. Consequently, it has been shown that it performs poor in flow situations such as flows with sudden changes in mean strain rate, flows over curved surfaces, flow in ducts, flow of rotating and/or stratified fluids, three-dimensional flows. Among these models, $k-\varepsilon$ model is the most widely applied two-equation model. Despite the adaptation to the constants of the model for atmospheric flow done by [Crespo et al., 1985] et al. $k-\varepsilon$ models are still too diffusive for wind energy applications. Better agreement with experiment was obtained with another model, the *Reynolds stress model* that does not directly utilize the *eddy-viscosity* concept of the Boussinesq hypothesis. Disadvantage within this model is that it brings
second-order closure relations, which contain so many unclosed terms themselves that must be modeled often in arbitrary ways. They are shown to perform better than models relying on Boussinesq hypothesis but are computationally much more expensive at the same time [McDonough, 2004].

LES models split the turbulence into two parts, the large-scale turbulence and the small-scale turbulence and treats them separately. It calculates only the large eddies while the small-scale turbulence is modeled using a sub-grid scale turbulence model. The fundamental basis of LES is that the large-scale eddies are directly affected by the underlying conditions and include most of the Reynolds stress so that they must be calculated while the small-scale eddies contribute less to the Reynolds stress so that they are less crucial. LES allows the use of much larger cell sizes in comparison with DNS and also much higher Reynolds numbers. A major disadvantage of LES is that one requires to have very fine mesh near solid surfaces as the eddy sizes tend to be as small as in DNS so that there is a limitation for Reynolds number. The method that is used to overcome this difficulty is to have a hybrid model, LES-RANS turbulence model. LES provides us with more realistic atmospheric behavior which in turn brings expensive computations if one deals with more than a single wind turbine [Réthoré, 2009].

1.4 Current approach

In this thesis work we employed RANS equations with \( k-\varepsilon \) turbulence model to model the flow around a wind turbine. The wind turbine rotor is represented with actuator disc model. The flow solver is a new wind turbine wake CFD code that has been developed at ECN. This code is a 2D CFD code solving the incompressible Navier-Stokes equations with a Finite Volume Method (FVM) with a staggered grid arrangement, using a second order spatial discretization scheme with a central approximation of derivatives. RANS simulations for the flow around a wind turbine are conducted on a horizontal
plane with atmospheric inflow conditions at the hub height of the turbine. The results are compared with ECN wind turbine test site Wieringermeer measurements at the hub height and simulation results from FarmFlow code, which was developed at ECN for simulating wind farms with multiple turbines and calculating the wind speeds and power output values for such a wind farm arrangement.

2 Discretization of RANS equations

2.1 Navier-Stokes equations

In the absence of source terms the incompressible Navier-Stokes equations read

\[ \nabla \cdot \mathbf{u} = 0, \]
\[ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \nabla \cdot (\nu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)). \]

The diffusion term,

\[ D(\mathbf{u}) = \nabla \cdot (\nu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)), \]

can be written in Cartesian coordinates in two dimensions as

\[ D(\mathbf{u}) = \left( \begin{array}{c} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{array} \right) \cdot \left( \nu \left( \begin{array}{cc} 2 \frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & 2 \frac{\partial v}{\partial y} \end{array} \right) \right). \]

In order to deal with turbulent flows, the dependent variables are split in an average and a fluctuation \((u = \bar{u} + u')\), and subsequently the Navier-Stokes equations are (Reynolds) averaged. The turbulent stresses \(\tau_{ij} = \bar{u}_i' u_j'\) that result from the non-linear convective term are modeled with the Boussinesq' hypothesis:

\[ \tau = \tau_{ij} = 2
\nu T \delta_{ij} - \frac{2}{n} \kappa \delta_{ij}, \]
where

\[ s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \]  

(12)

The term \( \frac{2}{n} k \delta_{ij} \), with \( n \) the number of dimensions (2 or 3), is necessary to make the trace of the modeled stress tensor the same as the trace of the original stress tensor (being \( 2k \)). It can be absorbed in the pressure term. With the Boussinesq’ hypothesis the effect of turbulence is modeled as an increased viscosity. This turbulent viscosity, \( \nu_T \), is a function of space and time. The Reynolds-averaged Navier-Stokes equations then read:

\[ \frac{\partial u_i}{\partial t} + (u \cdot \nabla) u = -\nabla p^* + \nabla \cdot ((\nu + \nu_T)(\nabla u + (\nabla u)^T)), \]  

(13)

where \( p^* = p - \frac{2}{n} k \delta_{ij} \) and all quantities should now be seen as (ensemble) averages.

2.2 \( k-\varepsilon \) model

A possibility to determine \( \nu_T \) is to use the \( k-\varepsilon \) model. The 'standard' \( k-\varepsilon \) equations read [Wilcox, 1993]:

\[ \frac{\partial k}{\partial t} + (u \cdot \nabla) k = \tau \cdot \nabla u - \varepsilon + \nabla \cdot ( (\nu + \frac{\nu_T}{\sigma_k}) \nabla k ), \]  

(14)

\[ \frac{\partial \varepsilon}{\partial t} + (u \cdot \nabla) \varepsilon = C_{\varepsilon 1} \frac{\varepsilon}{k} \tau \cdot \nabla u - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \nabla \cdot ( (\nu + \frac{\nu_T}{\sigma_\varepsilon}) \nabla \varepsilon ). \]  

(15)

Note that \( \tau \) and \( \nabla u \) are both tensors, their product should be taken element wise. \( \tau \) is given by:

\[ \tau = \nu_T (\nabla u + (\nabla u)^T). \]  

(16)

In Cartesian coordinates (2D) the \( k-\varepsilon \) equations read:

\[ \frac{\partial k}{\partial t} + u \frac{\partial k}{\partial x} + v \frac{\partial k}{\partial y} = \nu_T \left( 2 \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + 2 \left( \frac{\partial v}{\partial y} \right)^2 \right) \]  

\[ - \varepsilon + \frac{\partial}{\partial x} \left( (\nu + \frac{\nu_T}{\sigma_k}) \frac{\partial k}{\partial x} \right) + \frac{\partial}{\partial y} \left( (\nu + \frac{\nu_T}{\sigma_k}) \frac{\partial k}{\partial y} \right). \]  

(17)
Table 1: $k$-$\varepsilon$ standard model constants [Wilcox, 1993]

<table>
<thead>
<tr>
<th>$k$</th>
<th>$C_\mu$</th>
<th>$\sigma_k$</th>
<th>$\sigma_\varepsilon$</th>
<th>$C_{\varepsilon 1}$</th>
<th>$C_{\varepsilon 2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.40</td>
<td>0.09</td>
<td>1.00</td>
<td>1.30</td>
<td>1.44</td>
<td>1.92</td>
</tr>
</tbody>
</table>

\[
\frac{\partial \varepsilon}{\partial t} + u \frac{\partial \varepsilon}{\partial x} + v \frac{\partial \varepsilon}{\partial y} = C_{\varepsilon 1} \frac{\varepsilon}{k} \nu_T \left( 2 \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + 2 \left( \frac{\partial v}{\partial y} \right)^2 \right) - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x} \left( \nu + \frac{\nu_T}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x} \right) + \frac{\partial}{\partial y} \left( \nu + \frac{\nu_T}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial y} \right). \tag{18}
\]

$k$ and $\varepsilon$ determine $\nu_T$:

\[
\nu_T = C_\mu \frac{k^2}{\varepsilon}. \tag{19}
\]

2.3 Boundary conditions

In wind energy applications, the lower 200m or less of the atmospheric boundary layer is of interest, which is the atmospheric surface layer [Richards and Hoxey, 1993].

In general, boundary conditions should be set in such a way that in the absence of the turbines they should be able to provide a homogeneous boundary layer flow. The requirement for this is that the boundaries should be sufficiently distant from the turbines [Richards and Hoxey, 1993], which should be taken into account in specifying the domain and boundaries of the CFD simulation.

**Inlet Boundary:**

For a vertical rectangular flow domain the inflow of the computational model has the following velocity, turbulent kinetic energy $k$, and the dissipation $\varepsilon$ profiles [Richards and Hoxey, 1993]:

\[
u = \frac{u_s}{\kappa} \ln \left( \frac{z}{z_0} \right) \tag{20}
\]
Table 2: Inflow values for \( k \) and \( \varepsilon \) obtained from Equations 21 and 22

<table>
<thead>
<tr>
<th>( u_\infty ) [m/s]</th>
<th>5</th>
<th>7</th>
<th>9</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>( k/u_\infty^2 ) [-]</td>
<td>0.0041</td>
<td>0.0044</td>
<td>0.0047</td>
<td>0.0060</td>
</tr>
<tr>
<td>( \varepsilon/u_\infty^3 ) [-]</td>
<td>( 1.07 \cdot 10^{-4} )</td>
<td>( 1.20 \cdot 10^{-4} )</td>
<td>( 1.33 \cdot 10^{-4} )</td>
<td>( 1.91 \cdot 10^{-4} )</td>
</tr>
</tbody>
</table>

\[
k = \frac{u_*^2}{\sqrt{C\mu}}
\]

\[
\varepsilon(z) = \frac{u_*^3}{\kappa z}
\]

In the literature it is suggested that the assumption for a logarithmic velocity profile at the inlet is reasonable since that the terrain at the location of the inlet boundary is smooth. Otherwise, in the case of a complex terrain, this assumption would not give accurate results [Kasmi and Masson, 2010].

Note that the present code has a horizontal rectangular flow domain. In practice this means that the vertical profile is not fully required for the inlet boundary but only the \( k \), \( \varepsilon \), and velocity values at a certain height, thus it is the hub height in this case as stated previously.

In present work, we have retrieved friction velocity time-series from Wind Atlas data of ECN for the period of January 2005 until June 2009 in order to calculate \( k \) and \( \varepsilon \) values from them. The time series data is first binned for four velocity values: 5, 7, 9, and 11 m/s and friction velocity data is then averaged for these bins. In essence, there are four values of friction velocity that represent a wide range of atmospheric conditions, so there is no selection criteria for the stability condition of the atmosphere.

Consequently, the \( k \) and \( \varepsilon \) values listed in Table 2 are obtained using the formula 21 and 22.
At the sides, symmetry boundaries are set and at the outlet outflow boundary specified by the pressure is used.

Table 2 shows the values that have been calculated from the friction velocity time-series of Wind Atlas of ECN for four different velocities, and they are used later on as inflow conditions.

2.4 Actuator disc model

The incompressible Navier-Stokes equations in differential form with the inclusion of a body force read:

\[
\nabla \cdot \textbf{u} = 0, \\
\frac{\partial \textbf{u}}{\partial t} + (\textbf{u} \cdot \nabla)\textbf{u} = -\nabla p + \nabla \cdot (\nu(\nabla \textbf{u} + (\nabla \textbf{u})^T)) + \textbf{f}.
\]

As shown in the introduction chapter, the body force term in integral form, i.e. sum of the forces, equals the thrust of the rotor, thus it is given as \( T = C_T \frac{1}{2} \rho u_\infty^2 A_d \). After applying the non-dimensionalization with \( \frac{1}{2} \rho u_\infty^2 \), the body force in \( x \)-direction, \( \tilde{f} = f / \rho u_\infty^2 \), becomes in a discrete sense:

\[
\sum_j \tilde{f}_j \Delta y_j = \frac{1}{2} C_T A_d
\]

where \( A_d = D \) in 2D.

For a constantly loaded actuator disc: \( \tilde{f}_j^x = \frac{1}{2} C_T \).
3 Code verification

3.1 Lid-driven cavity

The lid-driven cavity flow has been the traditional test case for validation of Navier-Stokes codes and incompressible flow solvers since the work of Burggraf that was carried out in late 1960's [Burggraf, 1966]. The work of Botella et al. presents highly accurate benchmark results for the singular driven cavity flow at Re = 1000 obtained from a Chebyshev collocation method [Botella and Peyret, 1998]. The results for stream function, vorticity and pressure obtained from our code with the finest mesh of 256 by 256 are in very good agreement with the results presented by Botella as shown in Figures 4, 5, and 6. The pressure is set equal to zero at the center of the cavity. In Table 3, the velocity extrema are listed for various grid layouts, from a coarse grid with 16 by 16 to the finest with 256 by 256. Results from the latter present very good agreement with the benchmark results of Botella et al. [Botella and Peyret, 1998] and their plots are given in Figures 4, 5, and 6.

<table>
<thead>
<tr>
<th>$M = N$</th>
<th>$u_{min}$</th>
<th>$y$</th>
<th>$v_{max}$</th>
<th>$x$</th>
<th>$v_{min}$</th>
<th>$x$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Present</td>
<td>16</td>
<td>0.3021</td>
<td>0.1843</td>
<td>0.2941</td>
<td>0.8157</td>
<td>-0.4704</td>
</tr>
<tr>
<td></td>
<td>32</td>
<td>0.3604</td>
<td>0.1646</td>
<td>0.3536</td>
<td>0.8354</td>
<td>-0.5010</td>
</tr>
<tr>
<td></td>
<td>64</td>
<td>0.3816</td>
<td>0.1735</td>
<td>0.3709</td>
<td>0.8447</td>
<td>-0.5233</td>
</tr>
<tr>
<td></td>
<td>128</td>
<td>0.3867</td>
<td>0.1688</td>
<td>0.3755</td>
<td>0.8403</td>
<td>-0.5258</td>
</tr>
<tr>
<td></td>
<td>256</td>
<td>0.3882</td>
<td>0.1711</td>
<td>0.3766</td>
<td>0.8425</td>
<td>-0.5268</td>
</tr>
<tr>
<td>[Botella and Peyret, 1998]</td>
<td>160</td>
<td>0.3886</td>
<td>0.1717</td>
<td>0.3769</td>
<td>0.8422</td>
<td>-0.5270</td>
</tr>
</tbody>
</table>

Table 3: Extrema of velocity through the centerlines of the cavity. 2nd order. cosine mesh.

In order to have a finer mesh at regions that are close to boundaries a cosine-mesh layout is used (Figure 3). This allows us to have an increased level of accuracy in simulation results. For visualization purposes every third line of the grid is shown in Figure 3. The grid is finer when close to boundaries and coarser around the center of the cavity.
Figure 3: Cosine mesh layout

Figure 4: Contour plot of streamfunction $\psi$ for the lid-driven cavity flow at $Re=1000$. [Botella and Peyret, 1998]
Figure 5: Contour plot of vorticity $\omega$ for the lid-driven cavity flow at $Re=1000$.

Figure 6: Contour plot of pressure $p$ for the lid-driven cavity flow at $Re=1000$. 
The results of a mesh refinement study using the benchmark results of Botella et al. [Botella and Peyret, 1998] is given in Figure 7. Here we consider the benchmark results as \( f(x_k) \), true values, and \( y_k \) as the results from the simulations with grids of various mesh sizes.

Errors, also called deviations or residuals, can be formulated as:

\[
e_k = f(x_k) - y_k \quad \text{for } 1 \leq k \leq N.
\] (25)

There are several norms that can be used with the residuals, \( e_k \), to measure how far the simulation results data lie from the true values, \( f(x_k) \). Hence the following definitions for L1-norm and L2-norm errors are used in determining the errors.

L1-norm error, average error is defined as:

\[
E_1(f) = \frac{1}{N} \sum_{k=1}^{N} |f(x_k) - y_k|
\] (26)

L2-norm error, root-mean-square error is defined as:

\[
E_2(f) = \left( \frac{1}{N} \sum_{k=1}^{N} |f(x_k) - y_k|^2 \right)^{1/2}
\] (27)

In Figure 7 it can be seen that the rate of convergence when decreasing the size of the finite element mesh, \( h \), is approximating 2 that is in agreement with the discretization scheme. This mesh refinement study is based on the velocity extrema benchmark values along horizontal and vertical lines passing through the center of the cavity.
3.2 Actuator disc

In order to achieve a wind turbine wake model, the Navier-Stokes Equations should be solved with the help of Computational Fluid Dynamics (CFD). In this way wind flow through a wind turbine can be modeled. For doing so, there is a need for implementation of the wind turbine in the equations. One way of doing this is to put a body force acting against the flow. This has been done in CFD code developed at ECN. This code utilizes Finite Volume Method (FVM) of second order for incompressible Navier-Stokes Equations with staggered, non-uniform grid in 2D and general boundary conditions and body force. Its steady solver uses a mixed Picard/Newton method. Here, steady solver is used since the simulations are done for steady case. The staggered grid method implies that the velocities are at the cell faces and the pressure terms are at the cell centers. The pressure gradient terms are derived from pressure terms at the center so that the gradient terms are located at the places of the velocity terms. This helps prevent a possible decoupling, which is a cause for numerical oscillations, so called pressure wiggles. Next step would be to implement the steady state $k-\varepsilon$ turbulence model to this
code. Before starting the $k$-$\varepsilon$ model implementation to the code, some simulations of laminar flow through a wind turbine are conducted as a single actuator disc in a flow. Additionally, it is desirable to have finer grid near the actuator disc downstream and upstream in horizontal direction and on the nearby area of the actuator disc in vertical direction. This has been done through a modification on the mesh generation method for non-uniform grids. Types of boundary conditions chosen for this simulation are listed in Table 4. Various input parameters selected for the simulation are listed in Table 5.

Stretch factor is a parameter to transform the grid from a uniform one to a non-uniform one. Number of volumes in both directions, $x$ and $y$, specified as input parameters change regarding to this factor, e.g. in this simulation with a stretch factor of 1.02 resultant number of volumes in $x$-direction is 180 after the creation of a finer grid down- and upstream of the actuator disc as it gets coarser in far wake region. In this case the number of volumes in $y$-direction is 240 as the region with a width of $2D$ in vertical direction and the actuator disc at the center has two times finer grid.

The results from actuator disc model are in good agreement with the analytical solution as shown in Figures 10 and 11. The analytical solution for a lightly loaded actuator strip (in 2D) derived by Madsen reads [Madsen, 1988]:

$$p(x, y, \Delta p, R) = \frac{\Delta p}{2\pi} \left( \tan^{-1} \left( \frac{R - y}{x} \right) + \tan^{-1} \left( \frac{R + y}{x} \right) \right)$$  \hspace{1cm} (28)

$$v_x(x, y, \Delta p, R) = u_\infty - \frac{p(x, y, \theta, \Delta p, R)}{\rho u_\infty} - \frac{\Delta p}{\rho u_\infty} \text{ only in the wake}$$  \hspace{1cm} (29)

This analytical solution is valid for only lightly loaded cases, so we take $C_T = 0.01$. The relatively small discrepancies occur due to the set values at the boundaries, i.e. taking the values from the exact solution for setting the boundary values would yield less discrepancies; but the approach here was not as stated.
Table 4: Boundary conditions (BC) of the problem

<table>
<thead>
<tr>
<th>BC</th>
<th>Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_{left}$</td>
<td>Inflow</td>
</tr>
<tr>
<td>$u_{right}$</td>
<td>Outflow described by pressure</td>
</tr>
<tr>
<td>$u_{upper}$</td>
<td>Symmetry</td>
</tr>
<tr>
<td>$u_{lower}$</td>
<td>Symmetry</td>
</tr>
<tr>
<td>$v_{left}$</td>
<td>Inflow</td>
</tr>
<tr>
<td>$v_{right}$</td>
<td>Symmetry</td>
</tr>
<tr>
<td>$v_{upper}$</td>
<td>Outflow described by pressure</td>
</tr>
<tr>
<td>$v_{lower}$</td>
<td>Outflow described by pressure</td>
</tr>
</tbody>
</table>

Table 5: Input parameters

- *Reynolds number* $= 1 \cdot 10^3$
- Laminar flow, Steady solver
- $N_x = 180$
- $N_y = 240$
- $u_{inf} = 1$
- Smallest grid size in $x$-dir: 0.02
- Smallest grid size in $y$-dir: 0.025
- Stretch factor, $s_x$, for $x$-direction = 1.02
- Stretch factor, $s_y$, for $y$-direction = 2
- $C_T = 0.01$

Figure 8: Layout of the non-uniform grid
Figure 9: Contour plots of the velocity and the pressure for the flow through an actuator disc at Re=1000.

Figure 10: Analytical (Exact) and numerical (Present) solutions for $u$ in $x$ and $y$-directions.
Figure 11: Analytical (Exact) and numerical (Present) solutions for $p$ in $x$ and $y$-directions.

4 Results

4.1 ECN EWTW measurements

Since the end of 2002 the unit Wind Energy of ECN has made available the ECN Wind Turbine test site Wieringermeer, EWTW, which is located in the North East of the Province Noord-Holland 35 km eastwards of ECN Petten. This test site consists of test locations for turbines from the industry to conduct their turbine testing, and five Nordex N80/2500 wind turbines that are equipped for experimental research including a meteorological mast (met-mast) (Figure 12). The met-mast in the vicinity of the research wind turbines is labeled as met-mast 3 and is equipped with sensors at three different levels: 3D sonic anemometers at the heights of 109.1m, 80m, and 52m; Risøcup anemometers at 80m and 52m; wind vanes at 79.2m and 51.2m; and temperature, humidity, air pressure sensors at 78.4m. Additionally, all turbines have sensors for wind direction, the loads on the turbine and the power output. The five turbines are labeled with numbers 5-9, turbine 5 standing on the most left of the Figure 12 (most to the West). All turbines are the same model of Nordex, N80/2500 with a hub height of 80m, rotor diameter D of 80m, and rated power of 2.5MW. All are located equidistant to each other,
3.8D. As can be seen on Figure 12 met-mast 3 is subject to the wake of these turbines, especially of turbine 5 and turbine 6. The data that has been used in this work is from met-mast 3 within the period January 2005 – June 2009 and it is for the wake condition of met-mast 3, i.e. it is the wind speed and turbulence intensity measurements of sensors on met-mast 3 averaged for 10 minutes when northern winds are present such that the true wind direction is between $280^\circ$ and $60^\circ$ with respect to the North. Corresponding wind speed and turbulence kinetic energy measurements can be seen in Figures 13 and 14. Important to note are the trends that the wind speed deficits are higher in measurements for Turbine 6 that is 2.5D away from the met-mast 3, and that the crest be seen on the centerline angle for $11m/s$ case of Turbine 6. One possible explanation for this crest can be that the flow has less time for mixing while reaching 2.5D than 3.5D downstream, so that the presence of the hub as a low thrust region is visible. Furthermore, note that the profiles are asymmetric. If we look at the turbulence kinetic energy profiles we see again that those are not symmetric as well as two crests occur at the edges of the rotor possibly due to lower thrust near the hub of the turbines. Also, the dips and crests are more visible for Turbine 6, which is closer to the met-mast 3 than Turbine 5.
The layout of the EWTW is given in figure 1. The figure shows three rows of wind turbines where the upper two rows (i.e. the row denoted as 'Prototype turbines' and the row denoted as 'Research Turbines') form part of the EWTW. The middle row with prototype turbines is reserved for commercial testing of wind turbines. These turbines are numbered from 1 to 4. For the present study, the 5 research turbines at the northern part are of relevance. The research turbines are numbered from 5 to 9, with turbine 5 the most Westerly wind turbine. These turbines are variable speed, pitch controlled, and they have a diameter and a hub height of 80 m. The rated power is 2.5MW. Near the research farm, there is a 108 m high meteorological mast (which is denoted as M(et)M(ast) 3).

The distance and angles between the turbines and the meteorological mast are given in table 1 and figure 2. It can be noted that the wind farm line is directed 95-275 degrees with respect to the North. The distance between the turbines is 305 m (i.e. 3.81D). The meteorological mast is located at a distance of 282 m (3.5D) from turbine 5 under an angle of 315 degrees and at a distance of 201 m(2.5D) from turbine 6 under an angle of 31 degrees.

Table 1: Distance and angle between MM3 and the turbines

<table>
<thead>
<tr>
<th>Turbine</th>
<th>Distance [m]</th>
<th>Angle [deg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>T5</td>
<td>282</td>
<td>315</td>
</tr>
<tr>
<td>T6</td>
<td>201</td>
<td>31</td>
</tr>
<tr>
<td>T7</td>
<td>433</td>
<td>71</td>
</tr>
<tr>
<td>T8</td>
<td>722</td>
<td>81</td>
</tr>
<tr>
<td>T9</td>
<td>1022</td>
<td>85</td>
</tr>
</tbody>
</table>

(b) Research farm with five N80 turbines and the met-mast 3, distances and angles [Schepers, 2009]

Figure 12: EWTW map and layout of research farm
Figure 13: Wind speed measurements for Turbine 5 and 6

(a) Wind speed measurements for Turbine 5 in 4 wind speed bins

(b) Wind speed measurements for Turbine 6 in 4 wind speed bins
Figure 14: Turbulence kinetic energy measurements for Turbines 5 and 6
4.2 Comparison of RANS simulations with experiments and FarmFlow code of ECN

In order to run RANS simulations firstly we had to discretize the $k$-$\varepsilon$ equations. By implementing these equations into the code, we had the code ready for RANS simulations. As can be seen in Figure 15 the inflow conditions, e.g. the turbulence kinetic energy values at the inflow, are provided by the Wind Atlas of ECN, i.e. the turbulence kinetic energy and dissipation values for the hub height, 80 m, are calculated from the friction velocity time-series data of the Wind Atlas for corresponding wind speed bins. We have conducted RANS simulations for four different velocities: 5, 7, 9, and 11 m/s. The output of the CFD code is the pressure, velocity, vorticity, turbulence kinetic energy, and dissipation of the turbulence kinetic energy fields, and the streamlines. We then compared the results with the EWTW measurements and FarmFlow simulation results.

![Figure 15: Schema of RANS simulations](image)

Then for a rectangular and uniform grid, we have done a mesh refinement study to check the convergence of the results and the order of accuracy for the spatial discretization scheme, similar to the study performed for the lid-driven cavity.

The results of a mesh refinement study are given in Figure 16. Here we consider
Figure 16: Mesh refinement study
the results of the finest grid as the benchmark results, \( f(x_k) \), because there is neither an analytical solution nor benchmark results available for this case. The results from the simulations with grids of various mesh sizes are stated with \( y_k \) in error definitions.

Errors, also called deviations or residuals, can be formulated as:

\[
e_k = f(x_k) - y_k \quad \text{for } 1 \leq k \leq N.
\] \hspace{1cm} (30)

Similarly, there are several norms that can be used with the residuals, \( e_k \), to measure how far the simulation results data lie from the benchmark values, \( f(x_k) \). Hence the following definitions for L1-norm, L2-norm, and \( L_\infty \)-norm errors are used in determining the errors.

L1-norm error, average error is defined as:

\[
E_1(f) = \frac{1}{N} \sum_{k=1}^{N} |f(x_k) - y_k|
\] \hspace{1cm} (31)

L2-norm error, root-mean-square error is defined as:

\[
E_2(f) = \left( \frac{1}{N} \sum_{k=1}^{N} |f(x_k) - y_k|^2 \right)^{1/2}
\] \hspace{1cm} (32)

\( L_\infty \)-norm error, maximum error is defined as:

\[
E_\infty(f) = \max_{1 \leq k \leq N} \{|f(x_k) - y_k|\}
\] \hspace{1cm} (33)

The simulations for the mesh refinement study have been conducted on the same domain of 6 by 4 with systematically doubled number of mesh points, which are 60 by 40, 120 by 80, 240 by 180, 480 by 320, and 960 by 640. Similar to the situation visualized in Figure 17, which is the non-uniform grid that has been used later for RANS simulations that are compared with ECN measurements and FarmFlow results,
the readings for the mesh refinement study are taken from locations on an arc with a
distance of 2.5D from the disc. The locations on the two arcs in Figure 17 represent
the location of the met-mast 3 at varying wind inflow angles to the turbine within an
absolute range of 70°. As shown in Figure 17 the RANS simulations’ readings are taken
from locations on two arcs with distances of 2.5D and 3.5D from the disc to visualize
the conditions at the true met-mast location for varying true wind inflow angles. As a
result of the mesh refinement study with the mentioned rectangular uniform grid of var-
ious mesh sizes, the second coarsest grid gives quite close results of the velocity to
the results from the finest grid, which are shown in Figure 16 together with the results
of every different mesh size. The order of accuracy appears between 1 and 2; in fact
it was expected that it would turn out to be 2 as the spatial discretization scheme is of
2nd order but it was not the case; a possible explanation is that the actuator causes
discontinuities in the pressure as well as high velocity gradients at the edges due to its
uniform loading reducing the order of accuracy.

After the mesh refinement study we select the mesh size according to the second
coarsest grid that has been shown to be sufficiently fine for the RANS simulations as
shown in Figure 17, where only every fifth line is shown for a better visualization. Here,
we note that the grid is non-uniform, thus it is finer on a bandwidth of double the size of
actuator disc in y-direction and it is stretched in x-direction, finest around the actuator
disc and becoming coarser towards the domain boundaries. In order to cover most of
the downstream development of the wake, the outflow boundary is chosen to be further
away from the actuator disc while it is closer to the inflow boundary as it has less effect
on the flow. The span in terms of relative inflow angle is from −35° to 35°, which
 correspond to 280° to 350° with respect to the North for Turbine 5 that is 3.5D away
from met-mast 3 and −4° to 64° with respect to the North for Turbine 6 that is 2.5D
away from met-mast 3. The centerline from Turbine 5 to the met-mast 3 is at an angle
of 315° and the one from Turbine 6 to met-mast 3 is at an angle of 31°.
(a) Layout of the non-uniform grid and readings locations (Every fifth line is shown)

(b) EWTW layout

Figure 17: Non-uniform grid and EWTW layouts
Figures 18, 19, and 20 show velocity and pressure contours, vorticity contours and streamlines, and turbulence kinetic energy and dissipation contours, respectively. We note that the discontinuity in pressure due to the actuator disc is clearly visible as well the expansion of the wake visible in the streamline plot. Velocity contours indicate the overestimated wake expansion, thus the wake looks more diffusive when compared with laminar actuator disk results. Here, there are no sharp edges of the wake. The vorticity plot is in parallel with the turbulence kinetic energy and dissipation plots, where the vorticity created is the result of high gradients at the shear layer of the wake. Therefore turbulence kinetic energy is increased at the edges of the wake. In other words, the rotor extracts kinetic energy from the flow so that vorticity is created at the edges, which then is a reason for an increase in turbulence kinetic energy. Also, the turbulence kinetic energy is diffusing when we look further downstream, which indicates the mixing of free stream and the wake.

Figures 21, 22, 23, and 24 show the results for the wind speed and for the turbulence kinetic energy for Turbine 6 and 5, respectively; from our code, EWTW measurements and FarmFlow all together. First observation is that there is fair agreement between modeling results and the experiments, where both FarmFlow and our results show smaller velocity deficits than measured. Thus, the wake recovery is overestimated. However, the width of the wake seems acceptable as it is found by other researchers in the field such as Sanderse [Sanderse et al., 2011].

If we look at the turbulence kinetic energy results for Turbine 6 for example, all the three (measurements and results from two numerical simulations) show an increase in the turbulence kinetic energy. The higher velocity bins yield lower normalized turbulence kinetic energy values, which is comparable to “Turbulence intensity” while in absolute (dimensional) terms turbulence kinetic energy for higher velocity bins are higher. Crests are at about three-to-fourfold higher values than the inflow conditions. The tip vortices and wind shear at the edges of the wake of the rotor play a big role in creat-
Figure 18: Velocity and pressure contours
Figure 19: Vorticity contours and streamlines
Figure 20: Turbulence kinetic energy and dissipation contours

(a) Turbulence kinetic energy contours

(b) Turbulence kinetic energy dissipation contours
Figure 21: Velocity profile at 2.5D downstream of Turbine 6
ing turbulence. The dip occurs at the centerline where loads on blades are the lowest. Asymmetry can be recognized in measurements but not in the results from the simulations. FarmFlow shows the asymmetric behavior to the East of Turbine 6 as it covers the entire area with all the turbines whereas our simulations reflect only one actuator disc with a symmetric fashion of the domain and boundary conditions in 2D. Most importantly, FarmFlow is a parabolic code, which includes a prescribed near wake model, while the present model is fully elliptic and does not need such a prescribed near wake model.

![Figure 22: Turbulence kinetic energy profile at 2.5D downstream of Turbine 6](image)

We can state that the simulation results for Turbine 5 at 3.5D distance from the met mast 3 show better agreement with the measurements than the ones for Turbine 6 at 2.5D distance. Actuator disc assumptions and modeling assumptions cause discrepancies when we look at values at close distance to the actuator disc plane. We see better
results, i.e. close to the measurements, further downstream.

![Figure 23: Velocity profile at 3.5D downstream of Turbine 5](image)

The same trend is also visible for turbulence kinetic energy for Turbine 5. Two crests are visible at the edges due to velocity shear and the results are closer to the measurements for Turbine 5 than the ones for Turbine 6. In general the results show much better agreement with the measurements than FarmFlow.
Figure 24: Turbulence kinetic energy profile at 3.5D downstream of Turbine 5
5 Conclusions

• Simulations show reasonable agreement with experiments and Farmflow.
  - Turbulence kinetic energy is much better predicted than by FarmFlow.
  - Velocity deficit is approximately the same as FarmFlow.
  - There are promising developments for the use of $k-\varepsilon$ model for wind turbine applications. [Kasmi and Masson, 2008] showed that addition of new terms to $k-\varepsilon$ turbulence model for wind turbine applications and modified closure coefficients for $k-\varepsilon$ would yield better results.
  - One of the major advantages of RANS with $k-\varepsilon$ model with actuator disk is that the computation is relatively fast in the case of a reasonably small domain for a single actuator disc case.

• Discrepancies occur due to some assumptions involved within such an approach:
  - $k-\varepsilon$ turbulence model gives time-averaged flow field; in reality the flow field is highly turbulent and fluctuating. A more realistic turbulence model, like Large Eddy Simulation, is expected to give improved results, but is computationally more expensive.
  - 3D effects, e.g. wake rotation, wake meandering, 3D wake expansion, are not covered.
  - Temperature effects and ground effects are neglected in the simulations but are not filtered out in the measurements.
  - Uniformly loaded actuator disc causes major discrepancies at regions that are closer to the rotor.
References


