Hydrodynamics of horizontal-axis tidal current turbines

A modelling approach based on Delft3D

Master thesis

Sagar Mungar
Photo cover image:


Note: This document has been designed for full colour double-sided printing.
Hydrodynamics of horizontal-axis tidal current turbines

A modelling approach based on Delft3D

Master of Science Thesis

For obtaining the degree of Master of Science in Civil Engineering at Delft University of Technology

Sagar Mungar
Student number: 1396870

Faculty of Civil Engineering and Geosciences · Delft University of Technology
Abstract

A Horizontal-axis tidal current turbine (HATT) is a device that converts tidal energy into electrical power. The HATT uses the tidal current to generate electricity. There are plans to implement HATTs on a commercial scale in tidal farms similar to wind-turbine farms. However, experience with prototype farm arrangements in the field is lacking. This thesis focuses on the hydrodynamics of HATTs. The main areas of interest are the power production and hydrodynamic impact assessment of a tidal farm.

The HATT extracts momentum from the flow and as a result a wake forms downstream of the HATT. The wake can greatly influence the efficiency of HATTs positioned in the wake of another HATT, e.g. in tidal farms. HATTs may not only influence the efficiency of adjacent HATTs, but they may also affect the tidal motion on a large scale. The change of the tidal dynamics can in turn affect the power production of the HATT itself. The rotor diameter of HATTs are dimensioned to approximately half the water depth and in general HATTs convert 30 to 40% of the available energy in the current flowing through the rotor into electrical power. The relative large rotor diameter and high efficiency underline the hydrodynamic influence the HATT can have on the flow. A case study of a tidal farm conducted in the thesis shows that the hydrodynamic impact of a tidal farm can be significant and can extend to large areas.

The assessment of the hydrodynamic behaviour of shallow seas, in which HATTs are situated, usually requires the use of a numerical program which solves the shallow water equations. Therefore, it is interesting to know what the contribution of such a program can be to the hydrodynamic assessment of HATTs. In this thesis modelling of HATTs in a in such a program (Delft3D) and in theoretical models are compared to experimental data. The theoretical models use the self-similar properties of the wake to describe its evolution. From literature and the experiments it is known that the ambient turbulence is important for the recovery of the wake and is more influential compared to the rotor blades. Among other things, due to the large rotor diameter the influence of the free-surface and sea bottom on the wake is important, as the wake can be bounded by these boundaries.

The theoretical models are able to predict realistic values of the momentum extracted from the flow. At approximately $x/D = 7$ ($x$ being the coordinate in stream wise direction and $D$ the diameter of the rotor) downstream of the rotors the models predict the velocity within the desired accuracy of 15%. The accuracy of the theoretical models can be improved by setting more accurate boundary conditions and accounting for the influence of the free-surface and sea bottom.

Modelling of the HATT in Delft3D-FLOW focusses on large scale effects of the HATTs. The rotor of the HATT is modelled by a momentum sink. The momentum sink models the absolute momentum extraction from the flow. Delft3D is able to simulate the entire velocity field within the accuracy of 15% from approximately $x/D = 5$. Consequently, the velocity is predicted more accurately by Delft3D compared to the theoretical models. For locations different from the centre line of the wake, the desired accuracy is reached closer to the HATT, e.g. for $x/D < 5$. Overall Delft3D shows good potential to be used as a practical tool for the implementation of a tidal farm.
Preface

This report contains the Master Thesis work of Sagar Mungar. The Master thesis is part of the Master Civil Engineering at Delft University of Technology. The work was carried out in cooperation with the Hydraulic Engineering department of Deltares.

Acknowledgements
I would like to thank Deltares for giving me the opportunity to perform my Master thesis within the institute. The expertise and capabilities of Deltares in the field of hydraulic engineering have contributed greatly to the overall experience of writing the thesis.

Thanks to the members of my committee, Prof. dr. ir. W.S.J. Uijttewaal, Dr. ir. R.J. Labeur, Dr. ir. M.J.B.M. Pourquie and in particular to my daily supervisor, Ir. A.C. Bijlsma. Their insight, expertise and enthusiasm greatly contributed to the thesis.

Furthermore, I would like to thank the following persons for their support: Nick van der Bij, BSc. Maike Borst, Jathisan Chandrasegaram, BSc. Josh Friedman, Ir. Jelle van der Horst, BSc. Kelvin Ko, BSc. Edwin Lee, Suresh Sardjoe, BSc. Marta Skarżyńska and BSc. Jack Tai.

Finally, I would like to thank my parents, family and friends for their support during my studies in Delft.

Sagar Mungar
Delft, January 2014
# Contents

List of symbols.............................................................................................................................................. 1

1 Introduction .................................................................................................................................................. 3
   1.1 General introduction............................................................................................................................ 3
   1.2 Problem discription............................................................................................................................ 6
   1.3 Main question ......................................................................................................................................... 7
   1.4 Definition of scope ............................................................................................................................. 7
   1.5 General approach .................................................................................................................................. 8

2 Problem analysis ......................................................................................................................................... 9
   2.1 Power production ............................................................................................................................... 9
   2.2 Flow characteristics .......................................................................................................................... 10
   2.3 Theoretical modelling ....................................................................................................................... 19
   2.4 Conclusion ........................................................................................................................................... 28

3 Experimental data ................................................................................................................................... 29
   3.1 Introduction ......................................................................................................................................... 29
   3.2 Results .................................................................................................................................................. 32
   3.3 Comparison to theoretical models .................................................................................................... 37
   3.4 Conclusion ......................................................................................................................................... 41

4 Numerical modelling ............................................................................................................................... 42
   4.1 Introduction ......................................................................................................................................... 42
   4.2 Assumptions of Delft3D .................................................................................................................... 43
   4.3 HATT modelling ................................................................................................................................... 44
   4.4 Validation ........................................................................................................................................... 48
   4.5 Sensitivity analysis ............................................................................................................................. 60
   4.6 Conclusion concerning modelling of HATTs in Delft3D ................................................................. 73

5 Marsdiep case study ................................................................................................................................... 75
   5.1 Introduction ......................................................................................................................................... 75
   5.2 Tidal farm Marsdiep ............................................................................................................................. 76
   5.3 Expectations of the power production ............................................................................................... 79
   5.4 The model definition ........................................................................................................................ 79
<table>
<thead>
<tr>
<th>Symbol or abbreviation</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_{\text{DSS}}$</td>
<td>Quadratic friction coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$c_{\mu}$</td>
<td>Empirical constant</td>
<td>-</td>
</tr>
<tr>
<td>$V$</td>
<td>Gradient operator</td>
<td>-</td>
</tr>
<tr>
<td>$A$</td>
<td>Area</td>
<td>m²</td>
</tr>
<tr>
<td>$a$</td>
<td>Axial flow induction factor</td>
<td>-</td>
</tr>
<tr>
<td>$A_0$</td>
<td>Area of undisturbed flow</td>
<td>m²</td>
</tr>
<tr>
<td>$A_d$</td>
<td>Area at the actuator disc</td>
<td>m²</td>
</tr>
<tr>
<td>AEM</td>
<td>Algebraic Eddy viscosity closure model</td>
<td>-</td>
</tr>
<tr>
<td>$A_w$</td>
<td>Area in the far wake</td>
<td>m²</td>
</tr>
<tr>
<td>$B$</td>
<td>Buoyancy term or Blockage factor</td>
<td>j/s or -</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Power coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$C_t$</td>
<td>Thrust coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$D$</td>
<td>Diameter of the rotor</td>
<td>m</td>
</tr>
<tr>
<td>$D_k$</td>
<td>Diffusion coefficient for turbulent kinetic energy</td>
<td>m²/s</td>
</tr>
<tr>
<td>$D_s$</td>
<td>Diffusion coefficient for dissipation</td>
<td>m²/s</td>
</tr>
<tr>
<td>$F$</td>
<td>Force</td>
<td>N</td>
</tr>
<tr>
<td>$f$</td>
<td>Coriolis force per unit of mass</td>
<td>N/kg</td>
</tr>
<tr>
<td>$F_r$</td>
<td>Froude number</td>
<td>-</td>
</tr>
<tr>
<td>$H$</td>
<td>Hydraulic head</td>
<td>m</td>
</tr>
<tr>
<td>$h$</td>
<td>Water depth</td>
<td>m</td>
</tr>
<tr>
<td>HATT</td>
<td>Horizontal axis tidal current turbine</td>
<td>-</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy</td>
<td>m²/s²</td>
</tr>
<tr>
<td>$L$</td>
<td>Mixing length (turbulent length scale)</td>
<td>m</td>
</tr>
<tr>
<td>$M$</td>
<td>Momentum contribution due to external sources or sinks</td>
<td>m/s²</td>
</tr>
<tr>
<td>$O$</td>
<td>Order of magnitude</td>
<td>-</td>
</tr>
<tr>
<td>$P$</td>
<td>Pressure or power production</td>
<td>N/m² (Pa) or W</td>
</tr>
<tr>
<td>$P'$</td>
<td>Production term</td>
<td>j/s</td>
</tr>
<tr>
<td>$P_0$</td>
<td>Pressure just behind actuator disc</td>
<td>N/m² (Pa)</td>
</tr>
<tr>
<td>PD</td>
<td>Power density</td>
<td>W/m²</td>
</tr>
<tr>
<td>$Q$</td>
<td>Net average discharge</td>
<td>m³/s</td>
</tr>
<tr>
<td>$R$</td>
<td>Radius</td>
<td>m</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
<td>-</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
<td>s</td>
</tr>
<tr>
<td>$T$</td>
<td>Mixing time</td>
<td>s</td>
</tr>
<tr>
<td>$T_{T,s}$</td>
<td>Turbulent intensity in x direction</td>
<td>-</td>
</tr>
<tr>
<td>$TSR$</td>
<td>Tip speed ratio</td>
<td>-</td>
</tr>
<tr>
<td>$u$</td>
<td>Velocity component in the x-direction</td>
<td>m/s</td>
</tr>
<tr>
<td>$U$</td>
<td>Depth averaged velocity in the x-direction</td>
<td>m/s</td>
</tr>
<tr>
<td>$u_0$</td>
<td>Velocity of undisturbed flow</td>
<td>m/s</td>
</tr>
<tr>
<td>$u_1$</td>
<td>Velocity at centre line of wake</td>
<td>m/s</td>
</tr>
<tr>
<td>$u_2$</td>
<td>Bypass velocity</td>
<td>m/s</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>-------------------------------------------------------</td>
<td>---------------</td>
</tr>
<tr>
<td>$u_d$</td>
<td>Velocity at the actuator disc</td>
<td>m/s</td>
</tr>
<tr>
<td>$u_{df}$</td>
<td>Normalized velocity deficit</td>
<td>-</td>
</tr>
<tr>
<td>$u_w$</td>
<td>Velocity in the far wake</td>
<td>m/s</td>
</tr>
<tr>
<td>$v$</td>
<td>Velocity component in the y-direction</td>
<td>m/s</td>
</tr>
<tr>
<td>$V$</td>
<td>Depth averaged velocity in the y-direction or Volume</td>
<td>m/s or m$^3$</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Eddy viscosity</td>
<td>m$^2$/s</td>
</tr>
<tr>
<td>$w$</td>
<td>Velocity component in the z-direction</td>
<td>m/s</td>
</tr>
<tr>
<td>$x$</td>
<td>Spatial direction</td>
<td>m</td>
</tr>
<tr>
<td>$y$</td>
<td>Spatial direction</td>
<td>m</td>
</tr>
<tr>
<td>$z$</td>
<td>Spatial direction</td>
<td>m</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Width of wake</td>
<td>m</td>
</tr>
<tr>
<td>$\Delta t$</td>
<td>Time step</td>
<td>s</td>
</tr>
<tr>
<td>$\Delta x$</td>
<td>Size of grid cell in x-direction</td>
<td>m</td>
</tr>
<tr>
<td>$\Delta y$</td>
<td>Size of grid cell in z-direction</td>
<td>m</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Dissipation or dimensionless spatial direction</td>
<td>J or -</td>
</tr>
<tr>
<td>$\zeta$</td>
<td>Surface elevation</td>
<td>m</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Implicit factor</td>
<td>-</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Coefficient (value depends on definition of $\delta$)</td>
<td>-</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Velocity scale of turbulence</td>
<td>m/s</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
<td>m$^2$/s</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density of water</td>
<td>Kg/m$^3$</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Spatial direction</td>
<td>-</td>
</tr>
<tr>
<td>$\sigma_p$</td>
<td>Prandtl-Smidt number</td>
<td>-</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Shear stress</td>
<td>N/m$^2$ (Pa)</td>
</tr>
<tr>
<td>$\psi$</td>
<td>Dimensionless spatial direction in the z ($\psi = z/\delta$)</td>
<td>-</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Rotational velocity</td>
<td>Rad/s</td>
</tr>
</tbody>
</table>
Introduction

1 Introduction

1.1 General introduction
In the light of present day environmental concerns, the demand for clean sustainable energy sources is rising. A potential source of clean sustainable energy sources are the tides. Tides are long waves in oceans and seas, generated by the gravitational pull of the moon and the sun. Due to the rotation of the Earth around its axis, most places experience two high and low waters a day. The change from high to low water and vice versa induces a tidal current, see Figure 1 for an example. The predictable nature of the generating forces of the tides often results in a good predictability of the tides themselves. Tidal energy is not the only clean sustainable energy source, wind and solar power are other examples of such sources. Due to the predictable nature of the tides, they are especially attractive for power generation. A tidal energy device converts tidal energy into electrical power. The efficiency of this device is an important aspect for the economic viability of tidal power.

![Figure 1 Example of a tide and horizontal tidal current. The change of water level induces a horizontal tidal current. [2]](image)

There are several technologies available to extract energy from tides, such as vertical-axis tidal current turbines and horizontal-axis tidal current turbines. Horizontal-axis tidal current turbines (see Figure 2) referred to as HATTs for the remainder of the report, are the most mature and promising of the technologies [3]. Prototypes of HATTs are already operational in the field and there are multiple plans to implement HATTs on a commercial scale [3, 4].

A HATT works similarly as a horizontal axis wind-turbine. An important difference is the fact that the HATT is immersed in water and not air. The blades of the HATT convert kinetic energy from the tidal current into shaft mechanical energy. The power production of the HATT depends among other things on the velocity of the tidal current. Consequently, HATTs are applied in areas with large tidal currents. The minimum peak velocity for economical viable implementation of HATTs is 1 m/s [5].
Boundaries, such as land masses and the ocean topography distort the tidal wave, resulting in locally varying tidal characteristics which may result in relatively large tidal currents [7]. In short, shallow seas and land abutments tend to create environments which meet the current velocity criteria for HATT implementation. The number of sites in Europe with economically large enough tidal currents is limited. A possible location in the Netherlands to implement HATTs is the Marsdiep inlet [8]. Many favourable location are found in the seas around the UK, see Figure 3. It is estimated that approximately 20% of the power consumption in the UK could come from tidal power [9]. A prototype developed by Marine Current Turbine produces a peak power of 1.2 MW, which is enough to power 840 households\(^1\) [10, 11].

Tidal currents are not the only currents present in these waters. Storms, ocean currents and other type of currents may also be present, see Figure 4. The HATT is subjected to all of these local current conditions. These local currents are often smaller in magnitude and moreover occur less frequent compared to the tidal current. Thus, the latter is the dominant current in these waters. The tidal current is often\(^2\) bidirectional, e.g. it only fluctuates in direction with ebb and spring tides. It is expected that to a large extend HATTs with a fixed axis will be deployed, that is the turbines cannot adjust their axis to the current direction. Therefore, it is possible for the current to have a non-zero angle to the axis of the HATT, which is a difference to wind-turbines. The non-zero angle of the flow with the axis of the HATT influences the power output of the HATT.

---

\(^1\)Based on the average electricity use of Dutch households.  
\(^2\) Depends on local conditions.
Introduction

Figure 3 Peak flow for a mean spring tide U.K. [12].

Figure 4 hydrodynamic conditions for waters in which tidal turbines operate. [13]
A large rotor diameter $D$ is beneficial for the efficiency of the HATT. Consequently, the rotor of a HATT is dimensioned relatively large with respect to the water depth $h$. In practice this means that the rotors of HATTs are dimensioned to approximately half of the water depth $D \approx \frac{1}{2} h$ [14]. For example, in 30 m deep waters the diameter of a HATT would be 15 m. The large rotor diameter underlines the hydrodynamic influence the turbine can have on the flow, unlike wind-turbines whose rotor diameter is small with respect to the vertical length of the atmosphere. The boundaries (free surface and sea bottom) of HATTs are relatively close compared to wind turbines, implying that these boundaries can have a significant effect.

It is expected that HATTs will be applied in arrays, forming a tidal farm, similar to wind farms. Array formations are used to minimize the economic costs and to maximize the power production. The devices in tidal farms will be more closely spaced compared to those in most wind farms, as the largest current velocities are likely concentrated in a small area and the main current directions are fixed, in contrast to the conditions in most wind farms. There are some full scale HATT prototypes operating in the field at the moment of writing, however a full scale array formation of these turbines has not yet been implemented.

### 1.2 Problem description

The problem description presents the motivation for the research and serves as substantiation for the main question of the thesis.

In certain systems (for example the North Sea) tides behave as standing waves. Thus, in such a system the tidal wave can be regarded as a resonance phenomenon. Altering the characteristics of such a system (adding friction by applying a tidal power generation device) can alter the damping of the system, which may cause a change of the tidal amplitude. The change of the tidal amplitude can in turn affect the power production of the HATT itself. In general, the efficiency of HATTs is about 30-40%, meaning that 30 to 40% of the available energy in the current flowing through the swept area of the rotor of the HATT is converted into electrical power [11]. It is possible for tidal farms to significantly influence the tidal characteristics.

Tidal farms do not only influence the tidal characteristics but can have an influence on sedimentation and stratification as well. These influences can potentially cause large changes in the area of the tidal farm.

It is observed that downstream of a HATT a wake will form. A wake is defined as the free shear flow downstream of an obstacle [15]. The average velocity in the wake is lower than the average velocity of the undisturbed flow. In case of closely placed HATTs the wake of an HATT can (negatively) influence the power output of a HATT placed downstream, see Figure 5. This influence is more important for tidal farms than for wind farms because the tidal farms are likely to be more closely spaced than wind farms. It is likely that the wakes of individual HATTs will interact with each other, complicating the prediction of power production of a tidal farm.

---

3 HATTs are not dimensioned larger due to negative effects of the free surface and the seabed.
Introduction

The assessment of the hydrodynamic behaviour of shallow seas, in which HATTs are situated, usually requires the use of a numerical program which solves the shallow water equations. Therefore, it is interesting to know what the contribution of such a program can be to the hydrodynamic assessment of HATTs. A practical tool to assess the power production and impact of the HATT on the environment could be of great value. Especially, since there is no practical experience with tidal farms in the field yet.

![Figure 5 Schematical view of wake interaction. The inflow of the right turbine is determined by the wake of the left turbine. [11]](image)

1.3 Main question
The main question of the thesis is:

*What are the important flow characteristics involved in the implementation of horizontal-axis tidal turbines and how can these flow phenomena be simulated in a numerical model?*

The goal of this thesis is to determine if a numerical model (in this case Delft3D) can be used as a practical tool to optimize the energy production and assess the hydrodynamic impact of a tidal farm. But first, the flow phenomena induced by the turbine should be understood.

1.4 Definition of scope
The definition of scope elaborates on the aspects which are included or excluded from the thesis.

Because HATTs are the most mature and promising of tidal energy devices and there already are plans to implement these in the field, it is expected that HATTs will be the most commonly implemented tidal energy device [3, 4]. This thesis only focuses on the hydrodynamics and numerical modelling of HATTs.

The thesis includes the description of the influence of the free surface and the bottom (qualitative description). The influence of the angle of the current with the turbine, influence of waves, influence of
Introduction

marine biology growth, scour effects, other sedimentation effects and stratification effects are not included.

The main question includes the use of a numerical model. In this thesis Delft3D is used to model the flow phenomena. A case study, situated in the Marsdiep, based on Delft3D is conducted to demonstrate the value of a numerical program such as Delft3D.

1.5 General approach
The general approach elaborates on the method used to answer the main question. The general approach is explained on the basis of the contents of the Chapters.

- Chapter 2 contains the problem analysis. The problem analysis aims to provide essential theoretical information, tools to confidently tackle the problem described in the problem description. Different flow phenomena are identified and described in Chapter 2.
- To check the validity of the knowledge of Chapter 2, theoretical models are compared to experimental data. Chapter 3 presents the experimental data used to validate the theoretical models. Furthermore, the experimental data will be used as validation data for the numerical models in Chapter 4.
- Chapter 4 focuses on numerical modelling. The motivation for the used numerical model Delft3D is given and relevant assumptions of the program are discussed. Furthermore, attention is paid to the implementation of the HATT in the program and validation by comparison to the experimental data. Finally, a sensitivity analyses investigates the influence of several model settings on the accuracy of the program.
- Chapter 5 contains a case study. The general introduction mentioned Marsdiep as a potential site for HATT implementation in the Netherlands. It is investigated what the hydrodynamic impact of a tidal farm in Marsdiep is. Not all details will be included in the case study (such as stratification effects of the tide, morphological changes etc.), however the case study should show the general hydrodynamic impact of the farm. The case study shows the added value of a numerical program like Delft3D when planning a tidal farm.
- Chapter 6 consists of the discussion. The discussion elaborates on the errors and assumptions used to answer the main question.
- Chapter 7 is the final Chapter and contains the conclusions and recommendations. The chapter gives the outcome of the thesis and the recommendations for future work based on the findings and assumptions used in the thesis.
Problem analysis

2 Problem analysis

The problem analysis elaborates on the characteristics of the flow around the HATT. Chapter 2 forms the theoretical basis on which the other Chapters are built. The production of power by the HATT is a consequence of the interaction of the flow with the HATT. Therefore, attention is firstly paid to the power production and secondly to the relevant flow phenomena. Subsequently, the flow phenomena are described in more detail with theoretical models.

2.1 Power production

In general, the power production from a flow is calculated with the use of a dimensionless parameter, the power coefficient \( C_p \) [11, 16], see Equation 2.1.

\[
C_p = \frac{\text{electricity produced by device}}{\text{available energy in the flow}}
\]

The power coefficients of HATTs are typically about 0.3 to 0.4, meaning that 30 to 40% of the available energy in the current flowing through the swept area of the HATT is converted to electrical power\(^4\) [11]. The power coefficient depends on characteristics of the HATT and the available energy, more precisely it depends on the flow velocity and the rotational velocity and dimension of the rotor. The dimensionless parameter tip speed ratio \( \text{TSR} \) is introduced to be able to compare different HATTs. \( \text{TSR} \) is defined as:

\[
\text{TSR} = \frac{\omega R}{u_0}
\]

In which \( \omega \), \( R \) and \( u_0 \) are the rotational speed of the rotor, the radius of the rotor and the undisturbed velocity respectively. The power production is calculated with Equation 2.3:

\[
P = \frac{1}{2} \cdot \rho \cdot \int C_p \cdot u_0^3 \cdot dA \rightarrow P = \frac{1}{2} \cdot \rho \cdot C_p \cdot A \cdot u_0^3
\]

In which \( P, \rho, A \) and \( u_0 \) are respectively the power production, density of water, swept area of the rotor and the undisturbed velocity. Equation 2.3 assumes that the velocity and power coefficient are distributed uniformly over the swept area of the rotor. The swept area and the power coefficient are turbine characteristics and the density is a property of water, the value of these parameters can be found in literature. Consequently, if the velocity of the flow is known it is possible to calculate the power production. The velocity has a large influence on the power production as it appears to the third power in Equation 2.3. The importance of the flow velocity indicates the possible influence an upstream HATT can have on the power production of a HATT placed downstream. Studies of array formations of HATTs

\(^4\) The power coefficients of HATTs are of the same order of magnitude as wind-turbines.
such as Divett et al. [17] show that the efficiency of the tidal farm can significantly be influenced by the array formation. Divett et al. [17] conclude that array formations in which the turbines of alternating rows are placed directly between the upstream turbines have the greatest efficiency, e.g. a staggered array is more efficient.

Equation 2.3 is only applicable to a HATT from the cut-in velocity until the cut-out velocity. The cut-in velocity is defined as the undisturbed flow velocity from which the HATT begins to operate and the cut-out velocity is defined as the velocity from which the turbine shuts down for safety reasons. The stall range is defined as the velocity range where the turbine must be restrained. The cut-in velocity, cut-out velocity and stall range vary per manufacturer and type of HATT. Figure 6 gives an example of power production and power coefficient of a HATT.

![Figure 6 Example of the power production and power coefficient of a HATT. Modified from [18].](image)

2.2 Flow characteristics
The impact of the HATT on the flow is tangible upstream and downstream of the HATT. Firstly, attention is paid to upstream effects and secondly to the downstream effects. The flow characteristics are categorised as follows: inflow, wake, near wake and far wake.

2.2.1 Inflow
The HATT partly blocks the flow. A part of the flow is forced around the turbine with velocity \( u_2 \) and a part flows through the swept area of the rotor with velocity \( u_{\text{d}} \), see Figure 7. The bypass flow \( u_2 \) accelerates due to blockage of the flow. The acceleration of the flow can be explained by assuming a constant discharge in combination with a narrowing cross section. Continuity of the flow demands the
flow to speed up. The accelerated flow induces a deformation on the free surface, see Figure 7. In short, the HATT induces a resistance on the flow.

The flow passing through the swept area of the blades exerts a force on the blades. The force can be decomposed in a lift and drag force. The lift force is responsible for the rotation of the blades. In a well designed rotor the drag force is small compared to the lift force. The flow passing through the swept area of the blades performs work on the blades, the flow loses momentum. As a result of the momentum extraction the velocity of the flow passing through the swept area of the blades \( u_1 \) is smaller than the undisturbed flow \( u_0 \). The pressure in front of the turbine increases as the flow passes the rotor, momentum is extracted and the flow experiences a pressure drop. The flow exerts a resultant thrust force on the rotor as shown in Figure 7. The influence of the turbine on the inflow can extend to approximately 5 rotor diameters upstream of the turbine [19]. However, the biggest gradients are found in close proximity of the turbine.

![Figure 7 Schematic side view of a flow through a HATT. Modified figure from [20].](image)

The most important aspect of the inflow is the momentum loss of the flow. The thrust force on the turbine is a direct result of the momentum loss.

2.2.2 Wake
A wake is defined as a free shear flow downstream of an obstacle, see Figure 8. A free shear flow is a flow with mean velocity gradients that develops in the absence of boundaries. Consequently, a wake will form downstream of a HATT.

Free shear flows, and thus wake flows are characterised by small variations in the stream wise direction and much larger variations in the cross stream direction [15]. There is a sudden transition from an irrotational motion to a turbulent vorticity field, see Figure 8. The interface between the wake flow and ambient flow is highly convoluted. The characteristics of the interface change as a function of the Reynolds number (Re). The Reynolds number is defined as:
In which $D$, $u$ and $v$ are the characteristic length scale, the characteristic velocity and the kinematic fluid viscosity respectively. For Reynolds numbers smaller than $10^3$ the interface only undulates on a relative large scale, there is relatively little fine-scale turbulence. For Reynolds numbers larger than $10^4$ the interface shows both large and small scale undulations, the turbulence seems more developed and there are large- and fine-scale turbulence motions present in the flow [21]. The flow is regarded to be in the turbulent regime for Reynolds numbers above $10^4$. Figure 9 shows the wake behind a circular cylinder as a function of the Reynolds number. It is expected that the wake of a HATT depends in a similar way on the changes of the Reynolds number as the wake of the circular cylinder shown in Figure 9. Typical Reynolds numbers for waters in which HATTs are expected to be implemented are in the turbulent regime. Consequently, the wake of a HATT is classified as a turbulent wake.

![Planar jet and wake](image.png)

*Figure 8 Planar jet and wake. Modified from [21].*
Due to the rotational motion of the blades of the rotor, the wake of a HATT is more complex than the wake of a bluff body. The flow over a blade will form vortices at the trailing edge (back) of the blade. The flow cannot follow the rapid change in direction required to negotiate the sharp trailing edge, see Figure 10. Advection transports the vortices at the trailing edge in the downstream direction, these vortices are called shed vortices. Consistent with Newton’s third law, in reaction to the vortex at the trailing edge a vortex opposite in rotational direction will form: The bound lift generating vortex [23], see Figure 11. The bound lift generating vortex is, as the name implies, bound to the blade. Due to leakage of the flow from the high pressure area to the low pressure area at the tip of the blade the shed vortices roll up, see Figure 12. Note that the shed vortices of Figure 11 are represented by the vortex sheet in Figure 12. The vortex at the tip of the blade is called free tip vortex. The blades of HATTs unlike airplane wings have a twist along the length of the blades. The twist is necessary to ensure that the angle of the velocity vector with the blade chord (angle of attack) is relatively constant along the length of the blades.
Problem analysis

Figure 11 Schematic cross section of the wing, showing the bound vortex and shed vortex [24].

Figure 12 Tip vortex roll up. [25]

Similar to the wake of a wind turbine, it is expected that due to the symmetry of the rotor of a HATT an axi-symmetrical wake will form\(^5\). The wake develops similarly in lateral and vertical direction. Figure 13 shows the axi-symmetrical wake including the bound and free tip vortices.

Figure 13 Sketch of vortex systems behind a three bladed wind turbine [26].

\(^5\) In the case that the wake of the support structure of the HATT is neglected.
Problem analysis

The rotational motion of the blades causes extra turbulence compared to a wake of a bluff body and a non-zero swirl angle of the flow from the blades [27]. The swirl angle is the angle between the axial flow component of the velocity vector and the projection of the velocity vector on the surface defined by cross product of the axial and tangential component of the velocity vector [28], see Figure 14.

![Figure 14 Definition sketch of swirl angle. Modified from [29].](image)

Thus far, the nacelle (see Figure 15) has been neglected in the description of the wake. The nacelle acts as a bluff body. Consequently, the wake of the rotor consists of the wake of the blades as shown in Figure 13 plus the contribution of the nacelle. The support structure of the HATT also induces a wake. The total wake of the HATT consists of the combined wake of the rotor and the support structure.

![Figure 15 Schematic side view of rotor with nacelle. Modified from [30].](image)

It is expected that an axi-symmetrical turbulent wake will form behind a HATT. Note that the influence of the free-surface and bottom is not included in the description of the wake. In Section 2.2.4 attention is given to their influence on the wake.
2.2.3 **Near wake**

The expansion of the wake immediately downstream of the rotor is a result of the momentum extraction\(^6\). The tip vortices forms a cylindrical shear layer which separates the flow of the wake from the ambient flow, see Figure 16 [11]. The shear layer produces turbulence, the turbulence causes mixing of the ambient flow with the shear flow, consequently the width of the shear layer increases. The shear layer grows in the cross stream direction as shown in Figure 17 [11]. At approximately a distance of 3 à 4 rotor diameters D downstream the shear layer will reach the centre line of the wake [11, 31]. The downstream distance from the rotor to this point is called near wake, see Figure 17. Consequently, the near wake exists up until 3D à 4D downstream of the turbine [11, 31].

![CFD computer animation](image)

*Figure 16 CFD computer animation of a HATT showing tip vortices forming a shear layer and breaking up further downstream [11].*

Three minimum values of the velocity profile can be found directly behind the rotor, see Figure 17. The minimum velocity found in the middle is caused by the nacelle of the rotor. The other two velocity minima are caused by the blade geometry. Closer to the tip of the blade (the location of the two outer minima) the blades extract the most energy from the flow\(^7\) [11, 32, 33]. The minimum value of the velocity shifts to the centre line of the wake due to turbulent mixing within the shear layer, see Figure 17. The turbulence breaks down the tip vortices within the length of the near wake, see Figure 16.

‘The turbulence length scale, \(L\), is a physical quantity describing the size of the large energy-containing eddies in a turbulent flow’ [34]. The turbine generated turbulence will likely have turbulence length scales in the order of magnitude of the rotor diameter [35]. Bahaj et al. [31] states that the turbine generated turbulence will dissipate in the near wake. However, Batten et al. [27] and Roc et al. [36] have performed numerical simulations in which the turbine generated turbulence does play a role in the development of the wake. In short, the turbine generated turbulence plays a role in the development of

\(^6\)Decrease of velocity (momentum extraction) of the flow causes the area of the flow to increase (continuity), thus the wake expands. The density of water is considered constant.

\(^7\)It is beneficial for the power production to induce a large torque on the blades. Consequently, blades are designed to extract the most energy at an as large as possible distance away from the axis under the condition that structural integrity of the blade is not compromised.
the wake but the ambient turbulence is more influential turbulence parameter for the wake development.

The flow in the near wake can be characterised as complex, highly unsteady in nature and governed by detailed turbine characteristics.

![Figure 17 Schematic side view of the wake, shows expansion of the wake and velocity profile. Modified from [11].](image)

2.2.4 **Far wake**
In the far wake, the wake is considered fully developed, e.g. the minimum velocity can be found at the centre line and the width of the shear layers has reached the centre line, see Figure 17. Observation of wind-turbine wakes show that the minimum velocity downstream of the turbine is found below the centre line [11]. The downshift of the minimum velocity is a consequence of the proximity of the ground. The structure of the far wake is mainly maintained by convection and turbulent mixing. Note that the tip vortices are assumed to have been broken down in the near wake. Turbulent mixing transfers energy of the ambient flow to the wake flow causing the velocity deficit (the difference between the velocity in the wake and the velocity of the ambient flow) to decrease. The wake expands and recovers. The ambient turbulence intensity is an important parameter for the recovery of the wake.

The wake can no longer be distinguished if the velocity profile of the wake approaches the velocity profile of the undisturbed flow [31, 37]. The wake can exist up to large lengths, e.g. larger than 20D downstream of the turbine.

The velocity in the far wake is assumed to be independent on the detailed turbine geometry [37]. The thrust force and the turbine diameter are the only turbine parameters which influence the velocity profile in the far wake [11].

As the wake expands the influence of the free-surface and bottom increases [17, 38]. Experiments off HATTs show that the wake can be bounded by the free-surface and bottom [39]. Figure 18 shows the
vertical \((z)\) variation in the normalised velocity deficit of a wake. The normalised velocity deficit \(u_{df}\) is defined as:

\[
u_{df}(z) = 1 - \frac{u(z)}{u_0(z)}
\]

Equation 2.5

In which \(u(z)\) is the measured velocity in the flow direction and \(u_0(z)\) the undisturbed velocity. Figure 18 shows that the wake expansion is limited by the vertical dimension, e.g. at approximately 8\(D\) a 10\(D\) downstream the wake consists of the entire water depth. The influence of the free-surface and bottom distorts the wake causing a non axi-symmetrical wake to form [40].

![Figure 18 Vertical velocity deficit variation of a HATT wake (experiment) [39] development in downstream direction. D is the diameter of the rotor of the HATT. The free surface is located at \(z/D=0.8\) and the bottom at \(z/D=-0.83\).](image)

Wakes bounded in the vertical direction are also known as shallow wakes. Therefore, characteristics of shallow wakes are examined in relation to the wake of a HATT.

**Shallow wake**

A wake is classified as a shallow wake if the characteristic length scale of the object causing the wake is much larger than the characteristic length scale of the ambient flow, e.g. the Reynolds number of the
Problem analysis

wake \((Re = \frac{Du}{v}, D\) being the width or diameter of the object) is larger than the ambient Reynolds number \((Re = \frac{hu}{v}, h\) being the water depth).

The limited vertical dimension in shallow wakes give rise to the development of large scale two-dimensional structures [41]. ‘Horizontal vortical structures with a lateral dimension much larger than the flow depth are regarded as key features in the understanding of the transport of kinetic energy and mass in shallow wakes’ [41]. There are two mechanisms driving the large scale 2D motion: 1) a vertical shear layer introduced by bottom friction and 2) a horizontal shear layer caused by differences in lateral direction of longitudinal momentum transport [41]. The horizontal shear produces turbulence which is suppressed by the vertical shear layer. Bottom friction suppresses the generation of large scale motions and dampens their development.

Similar to a shallow wake, the wake of a HATT may form eddies which are larger than the water depth. These eddies stem from the lateral shear. The wake of a HATT can be considered a shallow wake if the entire water depth consists of the wake, e.g. the undisturbed velocity \(u_0\) is no longer present in the flow, see Figure 8. Further investigation is needed to determine when and if the wake of a HATT will form eddies which are larger than the water depth.

Conclusively, the wake of a HATT can be divided in three parts:

1. The near wake
2. The axi-symmetric three-dimensional wake (far wake)
3. shallow wake (far wake)

2.3 Theoretical modelling

The theoretical models give a more detailed description of the flow characteristics. The inflow is characterised by the momentum loss of the flow. The actuator disc theory elaborates on the momentum loss of the flow. The momentum loss subsequently induces a wake. Hence, wake models are presented which predict the velocity profile in a wake.

2.3.1 Actuator disc theory (Betz theory)

The actuator disc theory, also referred to as Betz theory, describes the principle of the momentum extraction independently of detailed rotor characteristics. The theory focuses on the key characteristics of the momentum extraction. It is regarded as the fundamental theory for the design and operation of wind-turbines and propellers [42]. The theory describes the momentum loss as a function of the thrust force on the turbine.

An actuator disc is a porous disc which is placed perpendicular to the undisturbed flow. The disc models the rotor of the turbine. It is assumed that the flow is incompressible, invicid, remote from boundaries and a steady state is considered. A control volume which satisfies the conservation of mass and does not allow mass transport through the boundaries is introduced, see Figure 19. There is a pressure drop over the disc, which manifests as a thrust force \(F\). Due to the pressure drop, momentum is extracted. The pressure is assumed to be uniform over the area of the disc. The pressure drop is followed by a decay of
Problem analysis

the velocity downstream of the disc. Consequently, the control volume expands to satisfy the mass conservation, see Figure 19 [11].

![Figure 19 Control volume with actuator disc, flow is from left to right [11]](image)

The force on the disc is calculated using the thrust coefficient $C_t$:

$$C_t = \frac{F}{\frac{1}{2} \rho \cdot A \cdot u_0^2}$$

Equation 2.6

In which $\rho$, $A$ and $u_0$ are the density, the cross sectional area of the disc and the undisturbed velocity respectively. The thrust coefficient is a dimensionless parameter which is a function of among other things the torque applied by the flow to the blades of the turbine. The torque is in turn a function of the flow velocity, the rotational velocity of the rotor and the diameter of the rotor. In short, the thrust coefficient is a function of the TSR, similarly as the power coefficient. The force on the disc is given by Equation 2.7.

$$F = \frac{1}{2} \cdot \rho \cdot \int \int C_t \cdot u_0^2 \cdot dA \rightarrow F = \frac{1}{2} \cdot C_t \cdot \rho \cdot A \cdot u_0^2$$

Equation 2.7

Equation 2.7 assumes the velocity and thrust coefficient to be distributed uniformly over the cross sectional area of the disc. The actuator disc theory supplies a relation for the velocity $u_d$ at the disc and for the velocity $u_w$ in the wake, see Figure 19. The velocities $u_d$ and $u_w$ are given by Equation 2.8 and Equation 2.9.

$$u_d = u_0 \cdot \left( \frac{1}{2} + \frac{1}{2} \cdot \sqrt{1 - C_t} \right)$$

Equation 2.8

$$u_w = u_0 \cdot \sqrt{1 - C_t}$$

Equation 2.9
Problem analysis

See appendix A for the derivation of these relations. The velocities are expressed as a function of the thrust coefficient and the undisturbed velocity. The momentum loss can therefore be calculated depending on the thrust coefficient of the turbine. The actuator disc theory describes the pressure distribution and momentum loss independent of detailed turbine characteristics.

Note that the actuator disc theory does not account for the proximity of the free-surface and the bottom. The free-surface proximity model elaborates on these influences.

**Free-surface proximity model**

The free-surface proximity model describes the effect of the proximity of the free surface and bottom on the relations described by the actuator disc theory. The free-surface proximity model is applicable to configuration of HATTs such as a linear array [20]. Basically, a two-dimensional vertical configuration is considered which does not allow for bypass flow \( u_2 \) (see Figure 20) left and right of the HATT. In advance one can state that this theory will overestimate the blockage effects, because in practice there is bypass flow left and right of the HATT. The free surface proximity model was introduced in Whelan et al. [20].

The bypass flow induces a height drop of the free surface as explained in section 2.2.1. The height drop and the performance of the disc are predicted by combining the continuity, Bernoulli and momentum equations. These equations are the base of the free-surface proximity model. The velocities are defined as:

\[
    u_w = \alpha u_0, \quad u_d = \beta u_0 \quad \text{and} \quad u_2 = \tau u_0, \quad \text{see Figure 20}
\]

Due to blockage of the flow, the bypass flow velocity \( u_2 \) will accelerate, see Figure 20.

Combining the continuity, Bernoulli and momentum equations results in a quartic polynomial of the change of velocity of the bypass wake flow \( \tau \) [20]:

\[
    Fr^2 \tau^4 + 4\alpha Fr^2 \tau^3 + (4B - 4 - 2Fr^2)\tau^2 + (8 - 8\alpha - 4Fr^2 \alpha)\tau \\
    + (8\alpha - 4 + Fr^2 - 4\alpha^2 B) = 0
\]

Equation 2.11

Figure 20 Control volume with actuator disc and boundaries. Modified figure from [20].
In which Fr = \( u_0 / \sqrt{gh_1} \) and the blockage factor B is defined as \( B = \frac{D}{h_1} \). See Appendix A for details of this derivation. By finding solutions for \( \tau \) in terms of \( \alpha \) it is possible to calculate various other parameters. The velocity values \( u_d \) and \( u_w \) found with free-surface proximity model differ from the values found with actuator disc theory. Based on Equation 2.11 it is possible to impose a correction to the thrust coefficient to account for the proximity of the free-surface and bottom, see Whelan et al. [20] for details.

The actuator disc theory and free-surface proximity model characterise the momentum loss of a HATT. In Section 2.3.2 theoretical models concerning the description of the wake are given.

2.3.2 Theoretical wake modelling

The theoretical wake models predict the velocity profile of the wake. There are several wake models available. These models are all based on the self-similarity principle of the wake. Therefore, the notion of self-similarity is clarified first. Secondly, the wake models of an axi-symmetrical wake, a shallow wake and Jenson wake model are presented. The self-similarity solution of the axi-symmetrical and shallow wake describes the behaviour of the wake in the ideal situation. By understanding the ideal situation first one can more easily understand complex situations. Of all the models tested by Palm [11], the Jenson model fits best to the wake of a HATT [11]. Due to the implementation of the HATTs in array formations it is likely that the wakes of individual HATTs will interact with each other. Therefore, the section concludes by presenting a modelling approach for the wake-wake interactions.

Self-similarity

A self-similar object is similar to a part of itself. ‘Scale invariance is an exact form of self-similarity where at any magnification there is a smaller piece of the object that is similar to the whole’ [43]. Scale invariance is the feature of an object or law that does not change if scales of length or other dimensions are multiplied by a common factor [43]. Self-similarity applied to wake flows embody that the velocity profile in each cross section of the wake is scaled in axial and radial direction, e.g. the velocity profile in each cross section has the same shape, see Figure 21 [11]. In Appendix B additional information concerning the notion of self-similarity can be found.
Problem analysis

Figure 21 Schematic view of an axis symmetric wake. The thick blue lines indicate the velocity profiles at different cross sections marked by the thin light blue lines. All profiles have exactly the same shape when scaled in horizontal and vertical direction.

Self-similarity requires properly scaled quantities in a wake to become independent of the scaled stream wise coordinate from a certain distance in stream wise direction [44]. In wake flows the velocity deficit and Reynolds stress are the quantities which become independent of the scaled stream wise coordinate [45]. Thus, these quantities constitute the self-similarity principle, see Equation 2.12 and Equation 2.13:

\[
\frac{(u_0 - u)}{u_1} = f(y/\delta, z/\delta) \quad \text{Equation 2.12}
\]

\[
\overline{-u'v'} = u_1^2 g(y/\delta, z/\delta) \quad \text{Equation 2.13}
\]

In which \(u_0\) is the undisturbed velocity in the (x) stream wise direction, \(u\) velocity in the stream wise direction, \(u_1\) is the minimum value of the velocity in the wake. The functions \(f\) and \(g\) depend on the normalized cross stream coordinates \((y/\delta\) and \(z/\delta\)) and \(\overline{-u'v'}\) is the Reynolds stress. The cross stream length scale \(\delta\) is defined as the distance from the centre line where the velocity deficit \((u_0 - u)\) is half the velocity deficit on the centre line of the wake, e.g. \((u_0 - u) = \frac{1}{2} \max |u_0 - u|\). See Figure 22 and Figure 23 for an illustration of the symbols. Substituting Equation 2.12 and Equation 2.13 in the equation of motion gives the self-similarity solution of the wake.
Self-similar solution of an axi-symmetrical wake

Substituting of Equation 2.12 and Equation 2.13 in momentum equations of an axi-symmetrical wake yields:

\[ \delta \sim x^{1/3} \]
\[ u_1 \sim x^{-2/3} \]

Equation 2.14

And velocity deficit \( u_{\text{dif}}(y) = u_1(x)\exp[-y^2/(\lambda \delta^2)] \)

In which \( \lambda \) is a dimensionless parameter which depends on the definition of \( \delta \). The derivation of the self-similar solution can be found in Appendix B. The self-similar solution describes and predicts the development of the width and the velocity profile of the wake. The solution is derived for the case in which there is no influence of the boundaries. The velocity profile of the self-similar solution appears to be similar to a Gaussian distribution, see Figure 24.
Problem analysis

Under the condition that the flow has had large enough downstream distance to evolve, Johansson et al. [46] finds the self-similar solution of the axis symmetrical wake \((\delta \sim x^{1/3})\) to be in excellent agreement with laboratory experiments and direct numerical simulations. Even though Johansson et al. [46] don’t specify the sufficient length, it can be stated that the solution is applicable to real wakes.

**Shallow wake**

For shallow wakes it is expected that as long as the transverse horizontal shear dominates the bottom friction, the wake shows the same dependency as a planar wake [41]. As the shallow wake develops, the relative influence of bottom friction will increase. The dependency is only valid in a limited range [41]. However, in order to be able to predict the behaviour of a shallow wake, the self-similar solution of a planar wake is investigated. The derivation of a planar wake is well known and can be found for example in Starke [44] and Davidson [21]. The self-similar solution for a planar wake yields:

\[
\begin{align*}
\delta &\sim x^{1/2} \\
u_1 &\sim x^{-1/2} \\
\text{And velocity deficit } u_{df}(y) &= u_1(x)\exp[-y^2/(\lambda \delta^2)] 
\end{align*}
\]

Equation 2.15

In Figure 24 an example of the self-similar solution of a planar wake (Equation 2.15) is shown. In the example \(\delta\) is defined as \(u_{df}(\frac{\delta}{2}) = 0.5u_1\) (which fixes \(\lambda=0.361\)) and \(\delta=14\) for \(x=0\). Figure 24 shows the development of the velocity deficit and the width of the wake \(\delta\).

![Figure 24 Example of self-similar solution of a planar wake, velocity deficit profiles drawn for downstream distance \(x=0, x=2, x=4\) and downstream \(x\) development of \(\delta\).](image)
Problem analysis

**Jenson model**
The Jenson model was originally derived for wind turbines. It neglects the near wake and assumes that the wake expands linearly ($\delta \sim x$), see Figure 25 [11]. The velocity in the wake is assumed to be uniform and is determined by the stream wise momentum equation, see Equation 2.16.

\[
p\delta_0^2 u_1 + \pi (\delta^2 - \delta_0^2) u_0 = \pi \delta^2 u_d
\]  Equation 2.16

A sinusoidal shape function is used to convert the uniform velocity distribution in the wake to fit to a Gaussian like distribution. Note, that the Jenson model requires calibration of the wake decay coefficient $k$ (see Figure 25). For additional information concerning the Jenson wake model see Palm [11].

![Figure 25 Jenson wake model. Modified from [11]](image)

**Wake interaction**
Due to the implementation of HATTs in array formations, interactions between individual wakes of HATTs are expected. This complicates the prediction of the velocity profile and subsequently the power output of a tidal farm.

![Figure 26 Schematic view of two HATTs and there corresponding wake (interaction) [11]](image)
Problem analysis

Generally, wake interactions are calculated with the superposition principle. The superposition principle states:

The net response of two or more stimuli is the sum of the responses which would have been caused by the individual stimulant.

There are multiple possible interpretations of the superposition principle [11]. One of the simplest superposition methods available, is the method which uses the maximum wake velocity as the dominant velocity, see Equation 2.17.

\[ u_{df} = \max \left\{ 1 - \frac{u_i}{u_0} \right\}_n \]

Equation 2.17

In which \( u_{df} \) is the normalized velocity deficit, \( u \) the velocity, the subscript \( i \) stands for the individual wake, \( u_0 \) is the undisturbed velocity and \( n \) is the number of wakes interacting. Figure 27 gives an example of the superposition principle. Palm [11] concludes that this method fits best to the wake interactions of HATTs.

![Figure 27 Development of planar self-similar solution of two wakes, shown is the normalized velocity profile (red and blue) in downstream position and their superposition (black).](image)

The wake models describe the velocity profile in the wake. The superposition principle applies the wake models to individual wakes to calculate the velocity profile of wake-wake interactions. The wake models and superposition principle can be used to describe the velocity profile of a tidal farm.
2.4 Conclusion
The inflow and wake of the HATT are the relevant flow characteristics. The most important aspect of the inflow is the momentum loss of the flow. The momentum loss manifests as a thrust force on the HATT. The wake can subsequently be divided in three parts:

1. The near wake
2. The axi-symmetric three-dimensional wake (far wake)
3. Shallow wake (far wake)

The flow in the near wake can be characterized as complex, highly unsteady and governed by detailed turbine characteristics. In the far wake, the velocity is assumed to be independent of detailed turbine geometry [37]. The thrust force and the swept area of the HATT are the only turbine parameters which influence the velocity profile in the far wake [11]. The ambient turbulence is more influential compared to the turbine produced turbulence and is an important parameter for the recovery of the wake. As the wake evolves, the influence of free-surface and bottom becomes larger [17, 38]. Experiments of HATTs show that the wake will be bounded by the free-surface and bottom. The wake of a HATT shows similarities to a shallow wake.

The actuator disc theory and the free-surface proximity model describe the momentum loss as a function of the thrust force on the turbine. The free-surface proximity model introduces a correction factor to include the influence of the free-surface and bottom. The wake models describe the velocity profile of a wake and are all based on the self-similarity principle. Self-similarity applied to wakes implies that the velocity profile in each cross section of the wake is scaled in axial and radial direction. Finally, the superposition principle allows for modelling of wake-wake interactions. With the help of the theoretical models one can predict the velocity profile and consequently the power production of a tidal farm.

In the next Chapter experimental data is presented. The experimental data will be compared to the theoretical models. It is investigated to what extent the theoretical models are valid.
In this Chapter the scale experiments of HATTs conducted by Stallard et al. [39] are discussed and compared to the theoretical models of Chapter 2. The experimental data presented in this Chapter is also used to validate the results of the numerical model in Chapter 4. Firstly, an introduction to the experimental data is given. Secondly, the results of the experiments are presented. Thirdly, the experimental data is compared to the theoretical models in. Finally, the conclusion on the experimental data is given.

3.1 Introduction
The goal of the experiments of Stallard et al. [39] is to investigate the wake of a HATT. The experiment focuses on the influence of the free-surface, bottom and proximity of other HATTs on the structure of the wake. The data of Stallard et al. [39] was selected as comparison data for the following reasons:

- Supplementary data of the experiments are public and free to use.
- The experiment focuses on the bounding effects of the free surface, bottom and adjacent wakes on the development of the wake. Several HATT formations are considered.
- The experiments have recently been published. The experiment continues on data collected in previous studies.
- The ratio of the rotor diameter over the water depth used in the experiments is similar to the ratio found in the field, hence the experiment represents a realistic situation.

Full details and results of the experiments can be found in Stallard et al. [39].

3.1.1 Experimental set up
The mean and fluctuating velocity components have been measured downstream of a three bladed HATT. The measurements have been performed on a single HATT and several formations of HATTs. The inflow conditions have been kept constant for every situation respectively. The HATTs are situated in a 5 m wide and 12 m long flume, see Figure 28 and Figure 29. The rotor diameter D is 0.27 m, the water depth 0.45 m and the mean velocity at z = 0 (the middle) is 0.47 m/s. ‘At 1:70th geometric scaling, these experiments represent a rotor diameter of 19 m, water depth of 31.5 m and applying Froude scaling a mean incident flow velocity of 3.93 m/s’ [39]. Note that Fr scaling is used in combination with high Re numbers to minimize scaling effects. The stream wise velocities have been measured to an accuracy of ±2% and the cross stream components to an accuracy of ±5%. It is assumed that the change of the tidal current velocity is slow. Consequently, the flow velocity in the flume is constant as, the steady state assumption is valid in this case, see Appendix C for details. The support structure consists of a 15 mm diameter tower which penetrates the free surface and has been designed as slim as possible. The tower extents to approximately hub height (z=0) and experiences a force in the order of 0.42 N due to the flow [39].
Figure 28 Arrangement of flume indicating key dimensions and global coordinate system. Three rotors and velocity measurement positions indicated for central rotor. Velocity measured at identical locations downstream of each rotor. Not to scale. [39]

Figure 29 Schematic three-dimensional view of the flume with the definitions of the axis, distances normalized by the diameter $D$ of the rotor.

3.1.2 Ambient flow
At the inflow a porous weir distributes the incoming flow and produces turbulence. Figure 30 shows the contours of undisturbed time averaged axial velocity $u$ at A) $x = 6$ m (corresponding to rotor plane) and B) $x = 7.5$ m (approximately $5.56D$ downstream of rotor plane). In general the velocity at positive $y$-axis flows faster than the flow at negative $y$-axis. The velocity field is not uniformly distributed. Figure 31 shows the undisturbed axial velocity $u$ at $x = 6$ m and $y = 0$ m. The vertical variation of the stream wise velocity does not increase monotone with decreasing water depth. It is important to notice that this non-uniform undisturbed velocity profile will distort the wake of the rotor.
Experimental data

Figure 30 Contours of axial time averaged velocity $u$ at A) $x = 6$ m (corresponding to rotor plane) and B) $x = 7.5$ m (approx. 5.56$D$ downstream of rotor plane). Note that the bottom is located at $z = -0.83D$. Data from Stallard et al. [39].

Figure 31 Vertical velocity profile present at $x = 6$ m and $y = 0$ m in the flume of Stallard et al. [39], bottom located at -0.83D.

The stream wise turbulence intensity $T_{1x}$ is presented\(^8\) as:

$$T_{1x} = 100 \times \sqrt[4]{\frac{(u - \bar{u})^2}{u_0}}$$

Equation 3.1

In which $u_0$ is the mean velocity upstream of the turbine, $\bar{u}$ is the mean velocity in the $x$-direction and $u$ is the measured (time varying) velocity in the $x$ direction respectively. The averaged ambient stream wise turbulence intensity\(^9\) at $x = 6$ m is about 10% (with maxima of 15%) and reduces to about 8% (with

---

\(^8\) In the original paper of Stallard et al. [39] there is no over bar indicating an ensemble average. It is assumed here that there is.

\(^9\) It is unclear how Stallard et al. [39] have computed values of averaged turbulent intensities. The averaged intensities of the cross section provided in the supplementary data of the experiments are 11.5% and 11.1% at $x = 6$ and $x = 7.5$ respectively, they do not correspond to the value mentioned by Stallard et al. [39].
Experimental data

maxima of 14%) at x = 9 m [39]. Based on the data of Osalusi [47] it is estimated that typical time averaged maximum turbulent intensities I (I is defined by considering the turbulence in all directions, e.g. I ≠ TIx) found in HATT implementations areas in the field are between 6% and 18%.

3.1.3 Turbine characteristics
In the experiments the turbine operates at a constant TSR. The expected power coefficient and thrust coefficient are 0.45 and 0.8 respectively [39, 48]. However, a thrust coefficient of $0.87 \pm 0.05$ is measured\(^{10}\). The deviation of the thrust coefficient is a result of the blockage of the flow [39, 48]. The blockage factor B is defined as the diameter of the rotor over the water depth, see section 2.3.1. In the case of these experiments is $B = 0.6$ and is considered as a high blockage factor. The high blockage factor illustrates the influence of the blockage potentially can have on the thrust coefficient.

3.2 Results
This Paragraph presents the measured velocities downstream of a single rotor, and three rotors by Stallard et al. [39]. The velocities are presented as the normalized velocity deficit $u_{df}$ (see Equation 2.5) and stream wise turbulent intensities $TIx$. The results show the characteristics of the wake of a HATT. Note that the normalized velocity deficit shows the absolute wake, e.g. wake of the HATT plus the support structure.

3.2.1 Measurements downstream of a single rotor
Figure 32 shows the normalized velocity deficit and stream wise turbulent intensity downstream of the axis of a single rotor. At $x/D < 4$ there are large gradients present, this region is classified as the near wake. At $x/D = 1.5$ the normalized velocity deficit is approximately 80 per cent, however at $x/D = 8$ the normalized velocity deficit recovers to about 20 per cent.

Figure 33 shows the lateral and vertical variation of the normalized velocity deficit and stream wise turbulence intensity at $x/D = 2$ downstream of a single rotor. The lateral normalized velocity deficit profile approaches a Gaussian distribution [39]. The maximum normalized velocity deficit is located below the centre line. A similar shift of the maximum velocity deficit is found in wind-turbine wakes, see section 2.2.4. The difference between the lateral normalized velocity deficit profile and vertical normalized velocity deficit profile implies that the wake is not axi-symmetric. This is likely caused by the influence of the free-surface and bottom.

\(^{10}\) The thrust coefficient is obtained by measuring force on the turbine and consequently calculating the thrust coefficient with the use of Equation 2.7.
Figure 32 A) Normalized velocity deficit and B) stream wise turbulence intensity downstream of the axis of single rotor.

Figure 33 Lateral and vertical variation of the normalized velocity deficit and stream wise turbulence intensity at x/D = 2 downstream of a single rotor. Data from Stallard et al. [39].
3.2.2 Measurements downstream of three rotors

In this section measurements downstream of three rotors placed side by side are presented, see Figure 34.

![Figure 34 Arrangement with three rotors, not on scale. Modified from [39].](image)

Figure 35 shows the normalized velocity deficit and stream wise turbulence intensity downstream of the axis of a single rotor and three rotors at 1.5D lateral spacing. Note that at x/D = 20 downstream of the rotors the normalized velocity deficit ranges from about 0.2 to 0.1. Consequently, the power production of a HATT placed 20D downstream is about 50% of the power production of the upstream HATT. The wake can significantly influence the power production of HATTs in array formations.

The centre rotor has the least amount of contact area, and consequently mixing with the ambient flow. Thus, the rate of recovery of the centre rotor is the slowest, e.g. the velocity deficit of the centre rotor shows the largest velocity deficit at x/D = 20 downstream of the rotors.

The case with three rotors does not produce significant additional stream wise turbulence compared to a single rotor. Consequently, one may expect that the rate of recovery of the wake of a single rotor is equal to the rate of recovery for three rotors. However, the normalized velocity deficit of a single rotor recovers faster\(^\text{11}\) compared to the normalized velocity deficit of three rotors. The explanation can be found by considering the lateral variation of the normalized velocity deficit. Figure 36 shows the lateral variation of normalized velocity deficit and stream wise turbulence intensity downstream of a row of three rotors at 1.5D lateral spacing. At approximately x/D = 5 downstream of the row of rotors, the wake of the individual rotors merge into a single wake. The merged wake has relatively less contact area with the ambient flow compared to the wake of a single rotor\(^\text{12}\). Accordingly, the normalized velocity deficit of a single rotor recovers faster.

Negative normalized velocity deficit values are found at the edges of the merged wake, see Figure 36. Negative normalized velocity deficit equals a velocity increase and is caused by the blockage of the flow. Note the asymmetry of the lateral variation of the normalized velocity deficit of the merged wake. The asymmetry in the normalized velocity deficit is caused by the asymmetry of the undisturbed velocity, see Figure 30 [39].

\(^{11}\) By faster it is meant shorter downstream distance in this case.

\(^{12}\) By simplifying the wake flow as a cone it is possible to understand that the merged wake has relatively less contact area with the ambient flow compared to the wake of a single rotor: When the size of the cone increases (representing merging of wakes in this case) the surface of the cone (base not included) will not increase as much as its volume.
Experimental data

Figure 35 A) Normalized velocity deficit and B) stream wise turbulence intensity downstream of the axis of a single rotor (circles) and three rotors at 1.5D lateral spacing: centre rotor y/D = 0 (blue curve), left rotor Y/D = +1.5 (black curve) and right rotor y/D = −1.5D (dashed curve). Data from Stallard et al. [39].

Figure 36 Lateral variation of A) normalized velocity deficit and B) stream wise turbulence intensity downstream of a row of three rotors at 1.5D lateral spacing. Data from Stallard et al. [39]. Note the differences in scale.
Experimental data

Figure 37 shows the lateral variation of the normalized velocity deficit at $x/D = 2$ downstream of three rotors lateral spacing 1.5D and three rotors lateral spacing 3D, including the velocity deficit of a single rotor. The unconstrained side of the normalized velocity deficit of three rotors laterally spaced by 1.5D is similar to the normalized velocity deficit of a single rotor, see Figure 37A. The normalized velocity deficit profile approaches the profile of a single rotor as the separation between the rotors increases. For 3D lateral spacing each wake is very similar to the single wake. Though asymmetry is observed in the outermost wake [39].

![Figure 37](image)

Figure 37 Lateral variation of the normalized velocity deficit at $x/D = 2$ downstream of three rotors lateral spacing 1.5D and three rotors lateral spacing 3D. The profile of isolated rotor wake also shown with filled circles. Data from Stallard et al. [39].

Figure 38 shows the vertical variation for the velocity deficit and streamwise turbulence intensity downstream of three rotors at 1.5D lateral spacing. The maximum normalized velocity deficit is found approximately 0.1D below the wake centre line [39]. The wake of three rotors is affected by the proximity of the free-surface and bottom, e.g. the wake is not able to expand in the vertical direction. At approximately $x/D = 8$ a 10 downstream of the rotors the wake consists of the entire water depth, similarly to the shallow wake.

Increased flow velocities are expected of the flow around the rotor. The vertical variation of normalized velocity deficit does not show increased flow velocities, see Figure 33 and Figure 38. Stallard et al. [39] states that increased flow velocities are observed at the upper and lower limits of the measurements range. Consequently, the measurements do not show the increased velocity.

---

13 to compare the results of a single rotor to multiple rotors, the centre line of the single rotor has been transposed to match the centre line of the experiments with multiple rotors.
In short, the wake can significantly influence the power production of HATTs in array formations. The wake appears to be non axi-symmetric. The free-surface and bottom bounds the wake in the vertical direction, e.g. the wake consists of the entire water depth. Finally, the wakes of multiple HATTs merge into one big wake.

### 3.3 Comparison to theoretical models

The theoretical models of Chapter 2 are compared to experimental data to check the validity of the knowledge of Chapter 2. Firstly, actuator disc theory is assessed and secondly the wake models are compared to the experimental data. Because the calculation of the power production of a HATT requires the velocity and not the velocity deficit, the velocities are compared and not the velocity deficits.

#### 3.3.1 Comparison to the actuator disc theory

The validity of the actuator disc theory is checked by comparing the calculated force on the rotor with the measured force on the rotor.

The force calculated with Actuator disc theory depends among other things on the thrust coefficient. Based on the expected thrust coefficient of 0.8 the force calculated with actuator discs deviates 9% from...
Experimental data

the measured force. This is a significant deviation. It is likely that it is caused by the influence of the free-surface and bottom. Consequently, the free-surface proximity model should be more accurate.

The free-surface proximity model requires detailed blade geometry to calculate the force. Unfortunately, these details are not available to the author. However, Whelan and Stallard [48] state that the force according to the free-surface proximity model is in the same order as the measured force.

The actuator disc theory and free-surface proximity model are able to predict realistic values for the thrust force acting on the HATT.

3.3.2 Comparison to the wake models
In this section the wake models are compared to the experimental data.

The self-similarity solutions of the axi-symmetrical and planar wake require that the velocity and width of the wake at a certain point, are specified. The self-similarity solution has been defined in two ways:

1. Defined by the measurement data of Stallard et al. [39].
2. Defined with the use of actuator disc theory

The use of actuator disc theory assumes that at x/D = 1 the width of the wake is one rotor diameter wide and the velocity is uniformly distributed at x/D = 1. The value of the velocity is provided by actuator disc theory.

In Figure 39 the velocity downstream of the axis of a single rotor is compared to the self-similar solution of an axi-symmetrical wake, planar wake and Jenson wake model. For self-similar solutions defined from other distances downstream with the data of Stallard et al [39] see Appendix J. At approximately x/D = 7 downstream of the rotor the self-similar solution of an axi-symmetrical and planar wake based on actuator disc theory are within 15% of the data of Stallard et al. [39]. The Jenson model does not model the velocity downstream of the axis very well compared to the other models. The relatively large deviation of the models to the experimental data in the near wake is due to the boundary conditions of the models and due to the limited range of applicability of the models. The self-similar solution of a planar wake defined by the actuator disc theory fits best to the values of Stallard et al. [39]. However, the rate of velocity recovery is slightly different. The self-similar solutions based on actuator disc theory are more accurate at x/D = 20 downstream of the rotor compared to the self-similar solutions based on the measurements of Stallard et al. [39].

The lateral and vertical velocities are compared to the data of three rotors. It is assumed that if the wake model is valid for three rotors, the model will be valid for a single wake as well. the wakes of three rotors include wake-wake interactions.

---

14 The self-similarity solution is only applicable in the far wake.
15 No information about the lateral and vertical variation of the velocity is available for a single rotor for x/D > 2.
Experimental data

Figure 39: Velocity downstream of the axis of a single rotor, with data of Stallard et al. [39], the self-similar solution of the axi-symmetrical wake, planar wake and Jenson model. The error bars indicate 15% deviation from the velocity.

Figure 40 shows the lateral velocity downstream of three rotors together with different wake models based on actuator disc theory. For the models defined by the data of Stallard et al. [39] see Appendix J. The data of Stallard et al. [39] does show self-similarity. Similar to the velocity downstream of the axis of the single rotor, the self-similar solution of a planar wake fits best to the data of Stallard et al. [39]. The superposition principle predicts the shape of the velocity profile well. It is important to notice that the models do not account for blockage of the flow. Consequently, the shape of the velocity profile of the theoretical models differ from the shape found in the experiments. Figure 40 clearly indicates that the velocity calculated with actuator disc theory is too high.

Figure 40: Lateral variation of the velocity downstream of a row of three rotors at 1.5D lateral spacing, with data of Stallard et al. [39] (blue), self-similar solution of axi-symmetrical wake (A, magenta), self-similarity solution of planar wake (B, cyan) and the Jensen wake model (C, red).
Experimental data

Figure 41 shows the vertical variation of the velocity, with data of Stallard et al. [39], self-similar solution of axi-symmetrical wake, self-similarity solution of planar wake and the Jensen wake model. The vertical shape of the velocity in the near wake is not modelled well by the theoretical models. Note that close to the bottom and free surface the theoretical models do not model the velocity very well as the theoretical models do not include the influence of the free-surface and bottom.

As expected, the wake models can only be applied a certain distance downstream of the rotor. At approximately $x/D = 7$ downstream of the rotors the models predict the velocity within 15% accuracy. Note that this is only valid in the case of the experiments of Stallard et al. [39]. The situation in the field is more complex (fluctuating flow velocity and direction etc.). The self-similar solution based on actuator disc theory of the planar wake fits best to the values of the experiments. The theoretical models are too simple in their prediction of the velocity shape, because the models do not contain influence of the free-surface and bottom.

![Figure 41: Vertical variation of the velocity with data of Stallard et al. [39], self-similar solution of axi-symmetrical wake (A, magenta), self-similarity solution of planar wake (B, cyan) and the Jensen wake model (C, green). Bottom located at $z/D = -0.83$.](image)
3.4 Conclusion
The conclusion differentiates the wake similarly as the flow characteristics of Paragraph 2.2: Near wake, far wake and shallow wake.

3.4.1 Near wake
Data from Chapter 2 suggests that the flow in the near wake is axi-symmetric, highly turbulent and governed by detailed turbine characteristics. High turbulence intensities and large gradients are found in the near wake. However, the data of Stallard et al. [39] shows that the near wake is not axi-symmetric. The non axi-symmetrical wake implies that the free-surface and bottom have influenced the wake to be non axi-symmetrical, the blockage of the rotor on the flow is significant.

3.4.2 Far wake
It is expected that the minimum velocity is found at or just below the centre line of the wake. The data of stallard et al. [39] is consistent with this expectation. The turbulence intensities in the far wake are significantly smaller than in the near wake. Most of the turbine generated turbulence is dissipated. The far wake is non-axis symmetric due to the non-axis symmetric near wake.

Wake models
The actuator disc theory and free-surface proximity model are able to predict a realistic value for the thrust force acting on the HATT. Furthermore, actuator disc theory is used to set the boundary conditions for the wake models. The velocity calculated with actuator disc theory is too high.

The wake of the rotor shows self-similar properties. Consequently, the wake of the rotor can be described by the self-similarity solutions. With the exception of the Jenson model, the self-similarity solutions based on actuator disc theory predict the velocity far downstream at x/D = 20 downstream of the rotor better than the self-similarity solutions based on the measurement data of Stallard et al. [39]. However, the self-similar solutions are too simple; blockage of the flow is not taken into account. The experiments of Stallard et al. [39] shows that the individual wakes merge into a combined wake. The combined wake is described by the superposition principle. Overall the far wake is consistent with knowledge from literature. At approximately x/D = 7 downstream of the rotors the models predict the velocity within 15% accuracy. Note that this is only valid in the relative simple case of the experiments of Stallard et al. [39].

3.4.3 Shallow wake
The far wake consists of the entire water depth at approximately x/D = 8 à 10 downstream of the rotors. There are no indications if eddies, which in horizontal direction are larger than the water depth, are formed. The normalized velocity deficit shows significant values at x/D > 20 downstream of the rotor. Hence, the wake can significantly influence the power production of HATTs in array formations.

In general the experimental data is consistent with the theoretical knowledge. The next Chapter focuses on modelling of the HATT in a numerical model.
4 Numerical modelling

The theoretical models can easily be applied to a simple case. However, for complex situations of actual tidal farms, it is more difficult to apply them. For complex situations a numerical model could be used to solve the velocity field. This Chapter elaborates on the modelling of HATTs in Delft3D. Paragraph 4.1 gives the motivation for the used numerical program Delft3D and the goal of the numerical model. The assumption made in Delft3D may not all be valid for the hydrodynamics of HATTs and so Paragraph 4.2 elaborates on relevant assumptions used in Delft3D. The HATT should be represented in the numerical model. Therefore, in Paragraph 4.3 attention is paid to modelling of the HATT. The method used to model the HATT is validated in Paragraph 4.4. When modelling a tidal farm, it is not always possible to use optimal settings to simulate the hydrodynamics of HATTs. Therefore, in Paragraph 4.5 the influence of model parameters is investigated. Finally, Paragraph 4.6 presents the conclusion of modelling HATTs in Delft3D.

4.1 Introduction

The Navier-Stokes equations combine the momentum and continuity equations to describe the motion of fluids. Modelling of all the length scales present in the flow with the Navier-Stokes equations, results in a relatively accurate prediction of the flow. However, for practical cases such an approach is unrealistic due to the large computational effort required. Therefore, simplifications are made to the Navier-Stokes equations to minimise the computational effort while trying to retain as much accuracy as possible. These simplifications often imply that a certain numerical model is only applicable to a limited range of flow problems [37].

4.1.1 Delft3D

Delft3D is a numerical software package developed by Deltares. The module Delft3D-FLOW is a part of Delft3D and is dedicated to the modelling of flows. Delft3D-FLOW is based on the shallow water equations which are a simplified version of the Navier-Stokes equations. ‘Delft3D-FLOW simulates unsteady flow and transport phenomena resulting from tidal and/or meteorological forcing, including the effect of density differences due to a non-uniform temperature and salinity distribution. Delft3D-flow can be used to predict the flow in shallow seas, coastal areas, estuaries, lagoons, rivers and lakes. It aims to model flow phenomena of which the horizontal length and time scales are significantly larger than the vertical scales’ [49]. The program is widely and successfully applied in numerous projects (see the website of Deltares www.deltares.nl).

In this thesis Delft3D is selected as the numerical program in which the hydrodynamics of a HATT are simulated, because it is already used to predict the flow phenomena in the areas where HATTs are expected to be implemented.

4.1.2 Goal

The goal of Delft3D is to assess the energy production and hydrodynamic impact of a tidal farm. It should serve as a practical tool for the implementation of HATTs. To reach the goal, Delft3D should
Numerical modelling

simulate the momentum extraction and the induced wake. In short, the flow in and around the tidal farm should be simulated. The main criteria of interest when modelling the HATTs are:

1. The accuracy to which the velocity of the wake and consequently, the interactions of the wakes can be modelled.
2. The accuracy to which the extraction of momentum from the flow can be modelled.
3. Finally, the accuracy to which the energy production of the turbine can be modelled.

It is not necessary for a practical tool to include all the details. Modelling of HATTs in Delft3D focusses on large scale (far wake) effects of the HATTs. Numerical models used as practical tools for wind farms show accuracies of 5 to 15% of the calculated velocities [11], this will also be the desired accuracy for Delft3D. In this case the calculated velocities refer to the velocity in the wake, not the velocity of the ambient flow. Note that due to the error propagation and errors in the ambient flow the accuracy of the power production can be more than 15%.

4.2 Assumptions of Delft3D

This Paragraph elaborates on relevant assumptions used in Delft3D-FLOW and their validity concerning the hydrodynamics of HATTs.

The shallow water equations have been derived from the Navier-Stokes equations by assuming that the vertical pressure is hydrostatic. Therefore, it is investigated if the hydrostatic assumption is valid concerning the hydrodynamics of HATTs. Secondly, attention is given to turbulence modelling, as turbulence plays an important role in the recovery of the wake.

4.2.1 Vertical acceleration

The hydrostatic assumption is valid if the vertical acceleration \( \frac{Dw}{Dt} \) can be neglected compared to the gravitational acceleration, e.g. \( \frac{Dw}{Dt} \ll g \). The vertical acceleration of the inflow just in front of the turbine in the case of Stallard et al. [39] is estimated to approximately:

\[
\frac{Dw}{Dt} \approx 1.20 \text{ m/s}^2 \tag{4.1}
\]

See Appendix D for the elaboration on the vertical acceleration. The vertical accelerations at the inflow are approximately 12% of the gravitational acceleration. Consequently, the hydrostatic assumption is not valid in close vicinity of the turbine. From literature, such as Carmer [41], it is known that for the near wake the hydrostatic assumption is not valid. However, for the far wake the hydrostatic assumption is valid [41].

The vertical pressures are by default assumed hydrostatic in Delft3D-FLOW. Nevertheless, Delft3D-FLOW has a non-hydrostatic mode to cope with flow phenomena in which the vertical accelerations cannot be neglected. This mode does take the vertical acceleration into account. See appendix D for a description of the non-hydrostatic mode.
Numerical modelling

4.2.2 Turbulence
The shallow water equations require a turbulence model to create a closed set of equations. This section elaborates on the modelling of turbulence.

Several turbulent models are developed, each specialised for a specific problem. Delft3D allows the user to choose between four different models, among which the k-ε model. The k-ε model has successfully been applied to free shear flows, including wakes [44, 50]. Therefore, the k-ε model is selected in this thesis. The k-ε model is based on transport equations for the turbulent kinetic energy $k$ and turbulent kinetic energy dissipation $\varepsilon$. See Appendix D for a detailed description of the k-ε model.

The k-ε model is applicable to a wide range of flows. The model is not specifically developed for wake flows and thus some discrepancies can be expected. For example, it is known that in general the k-ε model is slightly too dissipative in this case [51].

The vertical and horizontal turbulence are treated separately in Delft3D. See Appendix D for a detailed description of the incorporation of the turbulence models in Delft3D. Horizontal large eddy simulation (HLES) is a model for the horizontal turbulence. HLES can be used in combination with the three dimensional k-ε turbulence model. HLES assumes that the small scale turbulent motions are isotropic, e.g. they are not affected by large scale geometry. Modelling of large scale turbulent motions can be done on a relative coarse mesh\(^ {16}\). HLES simulates the unresolved small scale turbulent motions based on the horizontal mesh sizes, energy distribution in turbulence, Prandtl-Smith number (viscosity over diffusivity), gravitational acceleration, depth averaged flow velocity, water depth and friction and strain rates, for the exact description see Appendix D. The use of HLES can possibly improve the simulation of HATTs in Delft3D due to the additional turbulence term it introduces.

All the assumptions made in Delft3D can be found in the Delft3D-FLOW user manual [49].

4.3 HATT modelling
To assess the hydrodynamic impact and power production of a HATT in Delft3D, it is necessary to incorporate the HATT in the Delft3D model. This Paragraph elaborates on the representation of the HATT in the Delft3D model. Firstly, it is investigated if the HATT can be simplified and secondly, attention is given to the implementation of the (simplified) HATT in Delft3D.

4.3.1 Simplification of the HATT
Momentum extraction by a HATT can be modelled with the use of the actuator disc theory. This theory uses a porous disc to model the HATT. Delft3D has an option to define a porous disc. Therefore, it is investigated if the porous disc can model the wake of a HATT, e.g. if the HATT can be simplified to a porous disc.

\(^{16}\) The mesh is coarse compared to the length scale of the small scale turbulent motions. The mesh is not coarse, compared to length scale of the large scale turbulent motions.
Numerical modelling

The porous disc represents the effective swept area of the rotor and simulates the HATT by extracting momentum from the flow. The principle differences of using a porous disc, compared to an actual HATT are [31]:

- The momentum is extracted from the flow and not converted into the mechanical motion of the rotor.
- Vortices shed from the edge of the disc differ from vortices shed from the blades of a HATT.
- The swirl angle of the flow from the porous disc will be zero.

Bahaj et al. [31] state that these effects are exclusive for the near wake. The bulk of the turbine generated turbulence and the corresponding vortices are expected to dissipate in the near wake. It is assumed that the far wake is only influenced by the thrust, the diameter of the turbine, the ambient turbulence and, to lesser extent the turbine generated turbulence [11, 37], see Chapter 2. Hence, the far wakes of two geometrically different HATTs are similar if they have the same rotor diameter and same thrust force. Aubrun et al. [52] compared measurements of scale models of wind-turbines and porous discs. They conclude that the far wake of a wind-turbine is similar to the wake of a porous disc. Note that their conclusion is only valid for porous discs which induce a similar velocity deficit as the wind-turbine in the near wake (at x/D=0.5). Consequently, the porous disc can model the far wake if the disc models the thrust and diameter of the HATT.

Alternative to the modelling of the HATT as a porous disc, the HATT could be modelled on a method based on blade element theory. This theory describes the lift and drag forces on the blades by considering blade sections, it requires detailed blade geometry information.

Modelling of HATTs in Delft3D focuses on the large scale effects of the HATTs. Modelling of the near wake is of less interest. Thus, the HATT is modelled as a porous disc in Delft3D. Note that the support structure should be modelled separately. The next section elaborates on the implementation of the HATT simplified as a porous disc in Delft3D.

4.3.2 Implementation of the HATT

The porous disc is incorporated in Delft3D by the addition of an external momentum sink ($M_x$ or $M_y$) to the horizontal momentum equations, see Appendix D. An option to model the turbine generated turbulence is to prescribe turbulence to the porous disc [27, 36]. However, the focus is on the far wake and therefore no additional turbulence production term is prescribed at the porous disc. Note that in the future Delft3D could incorporate this extra turbulence term.

The external momentum sink as implemented in Delft3D has the form of an acceleration term:

$$M_x = c_{loss} \frac{|u_d| u_d}{\Delta x} \ [m/s^2] \quad \text{Equation 4.2}$$

$c_{loss}$ being a quadratic friction coefficient, $\Delta x$ thickness of the porous disc (the mesh size) and $u_d$ velocity at the disc. The quadratic friction coefficient is an input value for the definition of the porous disc. It is important to note that the implementation method of the porous disc in Delft3D may cause differences...
Numerical modelling

compared to actual physical porous discs. Differences between a physical porous disc and the porous disc as defined in Delft3D are among other things:

- The momentum extracted from the flow by a physical porous disc is converted into small scale turbulence downstream of the disc. In Delft3D the momentum is extracted by a momentum sink term and isn’t converted into small scale turbulence.
- The velocity inside the pores of the physical porous disc are likely larger than the average velocity at the porous disc, e.g. the physical porous disc allows the flow to vary locally. Whereas in Delft3D the flow only varies due to the uniform friction of the disc.

The quadratic friction coefficient of Equation 4.2 should be selected on the basis that the porous disc extracts the same amount of momentum from the flow as the HATT. As mentioned, the momentum loss manifests as a thrust force on the disc. Consequently, the force on the porous disc should equal the force on the HATT.

The force on the HATT is given by Equation 2.7:

$$ F = \frac{1}{2} \cdot C_t \cdot \rho \cdot \int u_0^2 \cdot dA \rightarrow F = \frac{1}{2} \cdot C_t \cdot \rho \cdot A \cdot u_0^2 $$  \hspace{1cm} \text{Equation 2.7}

The force on the porous disc is given by Equation 4.3:

$$ F = m \cdot a = \rho \cdot A \cdot \Delta x \cdot M_x \rightarrow F = \rho \cdot A \cdot c_{loss} \cdot |u_d|u_d $$  \hspace{1cm} \text{Equation 4.3}

m and a being the mass and acceleration respectively. Equating of the two forces leads to an expression for the quadratic friction coefficient:

$$ c_{loss} = \frac{1}{2} \cdot C_t \left( u_0 \right) \cdot \frac{u_0^2}{u_d^2} $$  \hspace{1cm} \text{Equation 4.4}

The determination of the quadratic friction coefficient depends on the thrust coefficient, undisturbed velocity and the velocity at the disc. Note that Equation 2.7 assumes the velocity and thrust coefficient are distributed uniformly over the cross sectional area of the disc. This assumption is passed on to the quadratic friction coefficient. In short, the absolute momentum extraction of the HATT is modelled, but the non-uniform distribution of the momentum extraction over the swept area of the rotor is not. The accuracy of Delft3D to model the far wake of the HATT may thus be influenced by the fact that only the absolute momentum extraction is modelled and due to differences between an actual physical porous disc and a porous disc as defined in Delft3D.

Note that it is still not clear how to obtain the value of the quadratic friction coefficient from Equation 4.4. The thrust coefficient $C_t(u_0)$ of a specific HATT is often provided by the HATT manufacturer. The difficulty in finding a value for the quadratic friction coefficient lies in the determination of the velocity at the disc. Three options to calculate the quadratic friction coefficient are investigated in Appendix E:
Numerical modelling

- With the use of the actuator disc theory (see Paragraph 2.3)
- With the use of the free-surface proximity model (see Paragraph 2.3)
- And by a calibration method.

Appendix E concludes that the calibration method models the momentum extraction the most accurate. Therefore, it has been selected as the method of HATT implementation. The calibration method is treated here, for a description of the methods based on actuator disc theory and free-surface proximity model see Appendix E.

**Calibration method**

The calibration method requires the user to specify a position from which a reference velocity \( u_{ref} \) will be taken. This reference velocity represents the undisturbed flow \( u_0 \). Recall that approximately 5 rotor diameters upstream of the HATT, the flow can be regarded as undisturbed\(^{17}\) [19]. The reference velocity is used to calculate the force on the disc, see Equation 4.5. This force is imposed on the porous disc, see Equation 4.6.

\[
F = \frac{1}{2} C_t \cdot \rho \cdot A \cdot u_{(ref)}^2
\]

\[
\frac{1}{2} C_t \cdot \rho \cdot A \cdot u_{(ref)}^2 = \rho \cdot A \cdot c_{loss} \cdot u_d \cdot u_d \Rightarrow \frac{1}{2} C_t \cdot u_{(ref)}^2 = c_{loss} \cdot u_d \cdot u_d
\]

The friction coefficient \( c_{loss} \) is adjusted in every time step so that the left hand side of Equation 4.6 equals the right hand side. As a result, the value of \( c_{loss} \) for every time step is found. The force on the disc equals the force on the HATT if the reference velocity is an accurate representation of the undisturbed velocity.

In tidal farms\(^{18}\), it is not always possible to assign a location from which the reference velocity can be taken. For cases, where it is not possible to select an appropriate reference velocity location, an additional calibration procedure relates the undisturbed velocity to the velocity at the disc. Consequently, the reference velocity can be taken at the HATT, e.g. \( u_{ref} = u_d \). For details of the additional calibration procedure see Appendix E.

**Post-processing: Power production**

Delft3D calculates the power production of a HATT with Equation 4.7:

\[
P = \frac{1}{2} \rho A u_{(ref)}^3 C_p
\]

The calculation of the power production is valid, if the reference velocity \( u_{ref} \) represents the undisturbed velocity \( u_0 \). Similarly to the momentum extraction, for cases where it is not possible to select an

\(^{17}\) Assuming that the HATT is the only disturbance on the flow.

\(^{18}\) Complex flows in which the undisturbed velocity cannot be used to calculate the thrust on the turbine.
appropriate reference velocity location, an additional calibration procedure can be used, see Appendix E.

A porous disc can model the far wake of a HATT if the disc models the thrust and diameter of the HATT. The HATT is modelled as a momentum loss term in Delft3D and a post-processing procedure calculates the power production of the HATT. The momentum loss term models the absolute momentum extraction from the flow. The accuracy of Delft3D to model the far wake of a HATT could be influenced by the fact that the non-uniform distribution of the momentum loss is not modelled and by differences between a physical porous disc and the porous disc as defined in Delft3D. In the next Paragraph the accuracy of Delft3D to simulate the far wake is investigated. For additional information about the implementation of the HATT in Delft3D see Appendix E.

4.4 Validation
The validation of Delft3D elaborates on the accuracy of Delft3D to simulate the hydrodynamics of a HATT. Therefore, the Delft3D model simulates and is compared to the data of Stallard et al. [39]. In section 4.4.1 the principle points to simulate the conditions of Stallard et al. [39] are discussed. Section 4.4.2 elaborates on the used model settings to simulate the conditions of Stallard et al. [39]. Finally, section 4.4.3 presents the results of the validation.

4.4.1 Conditions of the experiment
This section highlights the principle points required to simulate the conditions of the experiments of Stallard et al. [39].

1. The dimensions of the flume of the Delft3D model should be similar to the flume of Stallard et al. [39]. It is assumed that the walls and bottom of the flume of Stallard et al. [39] are made of glass.
2. The flow velocity and water depths must be similar.
3. The turbulent conditions must be similar.
4. Finally, the HATT characteristics must be similar.

The Delft3D model can be validated if the principle points described in the above description are similar to the experiments of Stallard et al. [39].

4.4.2 Used model settings of the validation
This section elaborates on the base settings used to validate the Delft3D model. The model settings are selected to accurately simulate the hydrodynamics of a HATT. Note that in the sensitivity analysis of Paragraph 4.5 the influence of various model settings are investigated.

Firstly, all the models used in the validation are computed in non-hydrostatic mode, see Appendix D for a description of non-hydrostatic mode. There are two ways to define the grid layers: sigma-layer model and Z-layer model. See Appendix D for a description of the grid layers. Non-hydrostatic mode is only available in the Z-layer model. Consequently, the Z-layer model is selected.
Numerical modelling

The enumeration in this section corresponds to the enumeration of the principle points of section 4.4.1.

1. The dimensions of the grid are similar to those of the flume of Stallard et al. [39], e.g. 5 m wide, 12 m long and 0.45 m in depth. Figure 42 shows the flume with the definitions of the axis. The horizontal mesh size is 0.093D and the vertical mesh size is 0.056D. Note that the use of a coarse mesh has more losses of extreme values compared to a fine mesh. A coarse mesh will transform steep gradients into less steep gradients, or in some cases will even average the gradients out as a whole. Furthermore, the time step is of less concern, due to the steady state considered in the experiments. The time step is only bounded by the restrictions set by the numerical scheme.

To simulate the (assumed) glass side walls of the flume, free slip conditions are used. For the (assumed) glass bottom a White-Colebrook bottom roughness coefficient of $5 \times 10^{-6}$ is used.

![Figure 42 3D schematic view of the flume with the definitions of the axis, distances normalized by the diameter D of the rotor.](image)

2. At the inflow boundary, a discharge of $0.0053 \text{ m}^3/\text{s}$ per horizontal grid cell is prescribed. This results in a depth average flow velocity of 0.47 m/s. At the outflow boundary, the water level is fixed to $z/D = 0.833$. This results in a total water depth of 0.45 m. The difference between the flow in the rotor free flume of Stallard et al. [39] and Delft3D models is shown in Figure 43. Note that the differences can be significant. Figure 43 is based on the model definition Delft3D_norotor, see Appendix G for full details.
3. In the validation cases the k-ε turbulence model is used to model the three dimensional turbulence, the HLES model is not used. Delft3D requires an input of the horizontal background viscosity, see Appendix D. The horizontal background viscosity is set to $1 \times 10^{-6} \text{m}^2/\text{s}$, the value is obtained from experience of normal values [49]. The data of Stallard et al. [39] lacks information about the cross stream (y and z direction) turbulence intensities. Therefore, it is assumed that the stream wise turbulent intensity accounts for 60% of the total turbulence intensity [53-55], see Appendix F. At the inflow boundary, turbulent boundary conditions have been imposed: The turbulent kinetic energy $k$ is set to 0.08 m$^2$/s$^2$ and corresponding dissipation $\varepsilon$ is set to 0.0576 m$^2$/s$^2$. This combination of $k$ and $\varepsilon$ leads to a turbulent viscosity of $10^{-2}$ m$^2$/s.

Table 1 shows that the stream wise turbulence intensity of Delft3D are similar to the data of Stallard et al. [39]. See Appendix F for elaboration of the conversion from turbulence kinetic energy to stream wise turbulence intensity.

<table>
<thead>
<tr>
<th></th>
<th>x= 6 m (x coordinate of the cross section)</th>
<th>x= 9 m (x coordinate of the cross section)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Averaged stream wise turbulent intensity at cross section of the Stallard et al. [39] experiments</td>
<td>10%</td>
<td>8%</td>
</tr>
<tr>
<td>Averaged stream wise turbulent intensity simulated by Delft3D</td>
<td>10.4%</td>
<td>8.7%</td>
</tr>
</tbody>
</table>

Table 1 Averaged stream wise turbulent intensities.

The turbulent boundary values are very high compared to the values at x = 6 m, see Figure 44. To have the correct ambient turbulent intensities at the location of the disc and consequently the wake it is necessary to impose these high turbulence boundary conditions. Studies such as Batten et al. [27] have shown that it is tolerated to have high turbulent intensities upstream of the disc. Appendix H shows the influence of the turbulent boundary conditions on the wake.
4. The diameter of the disc is 0.27 m and the quadratic friction coefficient has been calibrated to fit to a value of 0.87 of the thrust coefficient, see Appendix G. The reference velocity $u_{ref}$ is taken at the disc, e.g. $u_{ref} = u_d$.

For the full model definition details see Appendix G. It is expected that these settings will result in a relatively accurate simulation of the experiments of Stallard et al. [39]. The results of model definitions with these settings are presented in the next section.

4.4.3 Results

This section presents the results of the validation. Three cases are considered:

- A case with a single rotor in the flume.
- A case with three rotors in the flume.
- Finally, a case with five rotors in the flume.

The model definitions have been given a systematic name, for example Delft3D_3rotor3D means that it is a Delft3D-FLOW model which simulates 3 rotors laterally spaced by 3 rotor diameters.

**One rotor**

The model definition Delft3D_1rotor has all the model settings as described in section 4.4.2. Delft3D_1rotor is compared to the data of Stallard et al. [39] and the self-similar solution of the planar wake. For full model definition details see Appendix G.

The far wake of the HATT is similar to the far wake of a porous disc if the velocity deficit in the near wake is similar [52]. Thus, the lateral and vertical velocity profile at $x/D = 2$ downstream of the rotor of the experimental data is compared to Delft3D_1rotor. However, because the method of HATT implementation in Delft3D does not account for the distribution of the momentum loss and the near wake is governed by detailed turbine characteristics, the velocity profile at $x/D = 2$ cannot be directly compared. Instead of comparing the velocity profile, the absolute momentum loss of the experimental data at $x/D = 2$ is compared to the absolute momentum loss of the Delft3D model. Note that the
Numerical modelling

implementation method of the HATT in Delft3D ensures the correct\(^{19}\) amount of momentum is extracted from the flow (see Appendix E). Differences found in the absolute momentum loss at \(x/D = 2\) are thus a result of differences between extracting the same amount of momentum by a turbine compared to extraction of momentum by a momentum sink. Computational fluid dynamic (CFD) simulations could further clarify the differences between the two methods of momentum extraction.

Based on the velocity profile of the experimental data at \(x/D = 2\) (see Figure 33), the absolute momentum loss\(^{20}\) of the cross section can be estimated, see Appendix I. The absolute momentum of Delft3D_1rotor can be estimated similarly, see Appendix I. Delft3D_1rotor does not model the support structure, therefore it is expected that the absolute momentum loss of Delft3D_1rotor is smaller compared to the momentum loss of the experimental data. The force on the support structure is in the order of 0.42 N [39], see Chapter 3. Consequently, the absolute momentum loss of Delft3D_1rotor is expected to be about 0.42 N smaller than the experimental data. Note that the momentum loss can be expressed as a force, the results can be found in Table 2. It appears that the absolute momentum loss of Delft3D_1rotor at \(x/D = 2\) is much lower than the expected 0.42 N. The absolute velocity deficit of the Delft3D model is lower in the near wake compared to the experimental data. Consequently, at the start of the far wake (\(x/D ≈ 4\)) the velocity of Delft3D_1rotor will be too large.

<table>
<thead>
<tr>
<th>Momentum loss (force) of the experimental data</th>
<th>Momentum loss (force) of Delft3D_1rotor</th>
<th>difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.35 N</td>
<td>3.99 N</td>
<td>1.36 N</td>
</tr>
</tbody>
</table>

Table 2 Momentum loss (force) at \(x/D = 2\).

Figure 45 shows the longitudinal velocity and stream wise turbulence intensities downstream of the axis of a single rotor, with data of Stallard et al. [39], Delft3D_1rotor and self-similarity solution of the planar wake. At \(x/D = 4\) downstream of the rotor the velocity simulated by Delft3D_1rotor is larger than the velocity of the experimental data. Delft3D_1rotor is able to simulate the velocity within the desired accuracy of 15% from approximately \(x/D = 6\) downstream of the rotor. The stream wise turbulence intensity of Delft3D_1rotor doesn’t correspond to the stream wise turbulence intensities of the experimental data in the near wake, it does correspond from approximately \(x/D = 6\). The rate of the velocity recovery of Delft3D_1rotor only corresponds well to the experimental data if the stream wise turbulence intensities of both data sets correspond.

Delft3D_1rotor is more accurate compared to the self-similar solution of the planar wake, although the difference is small. The large difference between the self-similarity solution and to the experimental data in the near wake is due to the boundary conditions imposed on the self-similar solution (see Paragraph 3.3). The self-similar solution can be improved by using the experimental data as the boundary conditions for the self-similar solution, see Appendix J. However, if the experimental data is

\(^{19}\) According to Equation 2.7.

\(^{20}\) Note that for correct evaluation of the momentum loss the pressures should also be taken into account. Differences of the momentum loss due to differences in pressures are not taken into account in this case because one is interested in differences found in the velocity and not in the pressures.
Numerical modelling

used as the boundary condition for the self-similar solution, the self-similarity solution cannot be fairly\(^{21}\) compared to Delft3d model results. The rate of the velocity recovery of Delft3D_1rotor corresponds slightly better to the experimental data compared to the self-similar solution.

![Figure 45: Velocity and stream wise turbulence intensity downstream of the axis of a single rotor with data of Stallard et al. [39], Delft3D_1rotor and self-similarity solution of the planar wake. The error bars in the velocity indicate 15% deviation of the velocity.](image)

The accuracy of Delft3D can be improved by modelling the support structure. The model definition Delft3D_1rotor_support models the support structure and the rotor. The support structure of Stallard et al. [39] consists of a 1.5 cm diameter round cylinder. In Delft3D_1rotor_support the support structure is modelled by blocking the flow in the stream wise direction over a width of 2.5 cm. The influence of the support structure will be overestimated in this way, but Delft3D_1rotor_support should show the general influence of the support structure. For full details of Delft3D_1rotor_support see Appendix G. Note that the contribution of the nacelle has been accounted for, as the porous disc is calibrated according to the measured force of the rotor which includes the nacelle. However, due to the implementation method the contribution of the nacelle is spread over the entire area of the disc.

Figure 46 shows the velocity and stream wise turbulence intensity downstream of the axis of a single rotor with data of Stallard et al. [39], Delft3D_1rotor and Delft3D_1rotor_support. Note that due to the total blockage of the stream wise flow at the support structure, the stream wise velocity of Delft3D_1rotor_support will be zero at the support structure. The results of Delft3D_1rotor_support suggests, that the absence of the support structure in Delft3D_1rotor plays a role between differences

\(^{21}\) The self-similarity solution cannot honestly be compared because the Delft3D model does not use the velocity of the experimental data as input. The purpose of the models is precisely to predict this velocity.
Numerical modelling

in the minimum value of the velocity simulated by Delft3D_1rotor and the experimental data. The figure shows that at the beginning of the far wake (x/D > 4) Delft3D_1rotor_support is within the desired accuracy. Because Delft3D_1rotor_support overestimates the influence of the support structure it is reasonable to assume that Delft3D is able to simulate the velocity within the desired accuracy from approximately x/D = 5 downstream of the rotor. However, far downstream for example at x/D = 20 Delft3D_1rotor_support is less accurate than Delft3D_1rotor. It is likely that this is caused by the lower stream wise turbulence intensities found in Delft3D_1rotor_support. Note that the support structures of actual HATTs are expected to be larger than the support structure used in the experimental data. The support structure of full scale HATTs should not be neglected.

The accuracy of Delft3D can also be affected by the mesh resolution. Model definition Delft3D_1rotor_coarse4 has a horizontal mesh size which is four times coarser than the mesh size of Delft3D_1rotor. Figure 47 shows the velocity downstream of the axis of a single rotor with data of Stallard et al. [39], Delft3D_1rotor and Delft3D_1rotor_coarse4. The minimum velocity of Delft3D_1rotor_coarse4 is not as low as Delft3D_1rotor, see Figure 47. Based on the finding of Delft3D_1rotor_coarse4 it can be concluded that a finer mesh than used in Delft3D_1rotor would not greatly contribute to improvement of the accuracy.

\[22\] If the support structure of the experimental data is geometrically scaled.
Stallard et al. [39] did not provide data concerning the lateral and vertical variation of a single rotor for distances greater than x/D = 2 downstream of the rotor. Therefore, the lateral and vertical velocity profile shape will be validated for the case of three turbines. It is assumed that, if the lateral and vertical velocity profile is validated for three turbines, the lateral and vertical velocity profile of a single rotor is valid as well. The Figures with the lateral and vertical variation of the velocity and stream wise turbulence intensity of Stallard et al. [39] and Delft3D_1rotor at x/D = 2 can be found in Appendix J.

If the influence of the support structure is taken into account Delft3D simulates the velocity within the desired accuracy from approximately x/D > 5 downstream of the rotor. The inequalities of Delft3D_1rotor compared to the experimental data in the near wake can be related to the negligence of the non-uniform distribution of the momentum loss, differences between momentum extraction by a rotor and by a momentum sink term, due to the support structure and due to the influence of the discretization. The influence of the support structure of actual HATTs should not be neglected. The self-similar solution\(^{23}\) only reaches the desired accuracy from x/D > 7 downstream of the rotor. The velocity on the centre line of the wake of Delft3D_1rotor and the self-similar solution do not differ much for distances greater than x/D = 7 downstream of the rotor.

**Three rotors**

To investigate the capabilities of Delft3D to simulate wake-wake interactions, Delft3D is compared to the wake of three rotors. The rotors are placed side by side and are laterally 1.5D spaced. Delft3D_3rotor has all the model settings as described in section 4.4.2. For full model definition details see Appendix G. Model definition Delft3D_3rotor is compared to data of Stallard et al. [39] and the self-similar solution of the planar wake.

Figure 48 shows the longitudinal variation of velocity and stream wise turbulence intensity at the centre line downstream of three rotors at 1.5D lateral spacing with data from Stallard et al. [39] and Delft3D_3rotor. The velocity and stream wise turbulence intensity in the near wake of Delft3D_3rotor shows the same deviation from the experimental data as for the case of a single rotor. The velocity and stream wise turbulence intensity of Delft3D_3rotor corresponds within the desired accuracy to the

---

\(^{23}\) Note that the self-similar solution in this case does not account for the influence of the support structure.
Numerical modelling

experimental data from approximately $x/D = 7$ downstream of the rotors. The velocity on the centre line of a single rotor is modelled slightly more accurate by Delft3D than the velocity on the centre line of three rotors. Both the experimental data and Delft3D_3rotor shows that the recovery rate of the velocity of middle rotor is the slowest.

![Figure 48](image1.png)

Figure 48 Velocity and stream wise turbulence intensity downstream of the axis of three rotors at 1.5D lateral spacing. Data from Stallard et al. [39] and Delft3D_3rotor.

Figure 49 shows lateral variation of the velocity and stream wise turbulence intensity at sections downstream of a row of three rotors at 1.5D lateral spacing, with data from Stallard et al. [39], Delft3D_3rotor and the self-similar solution of a planar wake, note the differences in scale. The asymmetry in the merged wake of the experimental data is caused by the non-symmetrical ambient flow, see Chapter 3. The wakes of both the experimental data and Delft3D_3rotor merge and show similar widths. From $x/D = 7$ the entire horizontal velocity field is simulated within the desired accuracy. The accuracy can be improved by incorporating the influence of the support structure. Based on the findings of Delft3D_1rotor_support, it can be expected that the desired accuracy is reached from approximately $x/D = 5$ downstream of the rotor. Note that for distances smaller than $x/D = 7$ the velocity of Delft3D_3rotor in between the rotors is within the desired accuracy. As the wake evolves the area in which the velocity is resolved within the desired accuracy increases. The stream wise turbulence intensity of Delft3D_3rotor corresponds well to the experimental data from $x/D = 8$ downstream of the rotor. It is likely that the deviations in the near wake are caused by the absence of large velocity gradients, as found in the experimental data.
Numerical modelling

Figure 49 Lateral variation of A) velocity and B) stream wise turbulence intensity downstream of a row of three rotors at 1.5D lateral spacing. Data from Stallard et al. [39], Delft3D_3rotor and self-similar solution of the planar wake. The highlighted area in blue indicates deviation of 15% of the data from Stallard et al. [39]. Note the scale differences.

The wake of the self-similarity solution predicts the shape of the velocity profile well in the near wake except for the edges of the outer wakes. However, in the far wake the shape of the velocity profile of the self-similar solution does not correspond to the experimental data or the Delft3D model.

Figure 50 shows the vertical variation of velocity and stream wise turbulence intensity at sections downstream of a row of three rotors at 1.5D lateral spacing, with data from Stallard et al. [39], Delft3D_3rotor and the self-similar solution of the planar wake. The shapes of the velocity profile in the far wake of the three data sets seem to correspond. Note that at the free-surface and bottom the self-similar solution is no longer valid. From x/D >8 downstream of the rotors, the stream wise turbulent intensities of Delft3D_3rotor corresponds well to the experimental data, although deviation can be observed at the edges. It is likely that the deviations are caused by the variation of the ambient flow $u_0(z)$ in the flume of the experiments, see Figure 43.
Figure 50 Vertical variation of A) velocity and B) stream wise turbulence intensity downstream of three rotors at 1.5D lateral spacing. Data from Stallard et al. [39], Delft3D_3rotor and self-similar solution of the planar wake. Note the differences in scale.

Another model definition, Delft3D_3rotor3D, has a lateral spacing of 3D between three rotors, see Appendix G for full model definition details. Figure 51 shows the lateral variation of velocity of three rotors with 3D lateral spacing at x/D = 2 and x/D = 4 downstream of the rotors, with data from Stallard et al. [39] and Delft3D_3rotor3D. The wake of the outer rotor of Delft3D_3rotor3D is symmetric. However, the wake of the outer rotor of the experimental data is not symmetric. The high velocity deficit of the experimental data is likely the cause for the asymmetry of the outer wake. Delft3D_3rotor3D does not show this large velocity deficit. Consequently, the outer wake is symmetric.
Numerical modelling

Figure 51 Lateral variation of velocity of three rotors with 3D lateral spacing at $x/D = 2$ and $x/D = 4$ downstream of the rotors. Data from Stallard et al. [39] and Delft3D_3rotor3D.

Delft3D is able to simulate the shape of the merged wake well. The wakes of three rotors are predicted slightly less accurate than the wake of a single rotor. If the influence of the support structure is taken into account, it is expected that the entire velocity field is resolved within the desired accuracy from approximate $x/D = 5$ downstream of the rotor. The velocity in between the rotors is simulated within the desired accuracy for distances smaller than $x/D = 5$. The self-similarity solution does not simulate the shape of the velocity profile as accurate as the Delft3D model.

**Five rotors**

To investigate the capabilities of Delft3D to model more complex situations than three rotors, Delft3D is compared to a case with five rotors. The rotors are placed side by side 1.5D laterally spaced. The model definition Delft3D_5rotor has all the model settings as described in section 4.4.2. For full model definition details see Appendix G. Delft3D_5rotor is compared to data of Stallard et al. [39].

Figure 52 shows lateral variation of velocity at sections downstream of a row of five rotors at 1.5D lateral spacing, with data from Stallard et al. [39] and Delft3D_5rotor. From approximately $x/D = 7$ downstream of the rotor, Delft3D_5rotor simulates the velocity within the desired accuracy. Delft3D shows the same discrepancies of the lateral velocity profile of a row of five as to a row of three rotors. Delft3D is able to simulate the more complex situation of 5 rotors to the same accuracy as the case with three rotors. In short, the accuracy of Delft3D to resolve the velocity is not affected by the amount of wake-wake interactions.
Numerical modelling

Figure 52 Lateral variation velocity downstream of a row of five rotors at 1.5D lateral spacing. Data from Stallard et al. [39] and Delft3D_5rotor.

The validation shows that if the influence of the support structure is taken into account Delft3D is able to simulate the entire velocity field within the desired accuracy from approximately \( x/D = 5 \) downstream of the rotors. The support structure of actual HATTs play a significant role on the total wake. The velocity in between the rotors is simulated within the desired accuracy for distances smaller than \( x/D = 5 \). Thus, the velocity simulated by Delft3D is not resolved well on the centre line of the wake. Delft3D is able to predict the shape of the merged wake well. In the near wake the simulated stream wise turbulence intensities are lower than the values of the experimental data due to the absence of large velocity gradients in the simulated velocity. If the influence of the support structure is taken into account the Delft3D model performs better than the theoretical models. The velocity and the shape of the velocity profile is predicted better by Delft3D compared to the self-similar solution.

4.5 Sensitivity analysis

In Paragraph 4.4 the model settings are optimized to simulate the hydrodynamics of a HATT. However, when modelling a tidal farm it is not always possible to use the optimal settings. Therefore, in this Paragraph the influence of model parameters are investigated.
Numerical modelling

The influences of three model parameters are checked:

- The resolution of the mesh.
- The use of HLES.
- The use of hydrostatic mode.

Shallow seas in which the HATTs are expected to be implemented have horizontal length scales in the order of 100 km. Consequently, the model definitions used to simulate these shallow seas have a relative coarse mesh compared to the dimensions of the rotor of the HATT. The mesh size can locally be defined somewhat finer, in the order of 20 m. It is likely that HATTs will be modelled on these relative large mesh sizes. Therefore, it is important to know what the influence of the resolution of the mesh is.

The use of HLES can possibly lead to a better representation of the turbulent intensities and consequently, the resolved velocities. The effectiveness of HLES depends on the mesh size. Therefore, the influence of HLES is checked for various mesh sizes. It is important to note that the influence of HLES in this thesis is checked in combination with the Z-layer model and steady flow. However, the combination of HLES with the Z-layer model has not been fully tested [49] and due to the implementation method of HLES in Delft3D it is not effective for steady flows, see Delft3D user manual [49].

Hydrostatic mode is often used in the model definitions which simulate the shallow seas in which HATTs are expected to be implemented. It is checked if the hydrostatic assumption can be used to model HATTs. The use of hydrostatic mode enables the selection of two more model parameters:

- Grid layer model.
- Depth averaged flow.

In the validation cases the Z-layer model was selected, as non-hydrostatic mode is only available in Z-layer model. When hydrostatic mode is used, it is possible to use a single layer for the vertical mesh, this is referred to as depth averaged flow. Depth averaged flow is often used to reduce computational time. Figure 53 shows an overview of the investigated model parameters of this Paragraph.

---

24 The horizontal length scale is much greater than the vertical length scale in shallow seas. Consequently, from the conservation of mass it follows that the vertical velocity of the fluid is small. Thus, the hydrostatic mode is used to model the shallow seas.
4.5.1 **Mesh resolution**

Model definitions with various mesh sizes are compared to the data of Stallard et al. [39] in this section. The model definitions and their corresponding mesh size can be found in Table 3. Model definitions in which the rotor diameter of the HATT is smaller than the mesh size are called sub-grid models. In all the models 31 vertical layers have been used with exception of Delft3D_1rotor_subgrid. Delft3D_1rotor_subgrid has 15 vertical layers which vary in size, for full details see appendix G.

<table>
<thead>
<tr>
<th>Model definition</th>
<th>Mesh size ((\Delta x, \Delta y, \Delta z)) normalized by the rotor diameter (D (D=0.27 \text{ m})).</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delft3D_1rotor</td>
<td>(0.093, 0.093, 0.056)</td>
</tr>
<tr>
<td>Delft3D_1rotor_coarse2</td>
<td>(0.185, 0.185, 0.056)</td>
</tr>
<tr>
<td>Delft3D_1rotor_coarse4</td>
<td>(0.370, 0.370, 0.056)</td>
</tr>
<tr>
<td>Delft3D_1rotor_Hsubgrid</td>
<td>(3.704, 3.704, 0.056)</td>
</tr>
<tr>
<td>Delft3D_1rotor_subgrid</td>
<td>(3.704, 3.704, 1)</td>
</tr>
</tbody>
</table>

Table 3 Model definitions and their corresponding mesh size.

Figure 54 shows the longitudinal velocity and stream wise turbulent intensity on the centre line of single rotor with data of Stallard et al. [39] and various Delft3D model definitions with varying mesh sizes. The differences between the non-sub-grid models are small. At approximately \(x/D \approx 7\) downstream of the rotor, the velocity at the centre line of all the model definitions correspond to the experimental data within the desired accuracy. Large gradients of the near wake require high grid resolutions. Consequently, the influence of the mesh size is mostly limited to the near wake. Note that it is likely that the incorporation of the support structure in the sub-grid models would not result in a large contribution to the accuracy as it would for the non-sub-grid models. The turbulence intensities of the sub-grid models show that the mesh is too coarse to simulate the turbulence induced by the porous disc. As a result, the turbulent intensities in the far wake remain lower. Note that upstream of the disc the stream wise turbulence intensities of the sub-grid models are larger than the non-sub-grid models. This is caused by lower dissipation rates of the turbulence boundary conditions of the sub-grid-models. The dissipation rates of the sub-grid models are lower due to coarse meshes of the sub-grid models.

The lateral and vertical velocity and stream wise turbulence intensities of Delft3D are compared to the experimental data for the case with three rotors, because the experimental data lacks this information for the case of one rotor.
Numerical modelling

Figure 54 Velocity and stream wise turbulence intensity downstream of the axis of a single rotor with data of Stallard et al. [39] and various Delft3D model definitions with varying mesh sizes.

Figure 55 shows the lateral variation of velocity and stream wise turbulence intensity at sections downstream of a row of three rotors at 1.5D lateral spacing with data from Stallard et al. [39] and various Delft3D model definitions with varying grid sizes. Similarly to the previous sections, from x/D = 7 downstream of the rotor the velocity of all the delft3D models corresponds to the experimental data within the desired accuracy. However, the shape of the velocity profile of the sub-grid models doesn’t correspond to the velocity profile of the experimental data. The non-sub-grid model definitions do simulate a similar velocity profile as the experimental data. The turbulence induced by the porous disc is not modelled by the sub-grid models.
Numerical modelling

Figure 55 Lateral variation of A) velocity and B) stream wise turbulence intensity downstream of a row of three rotors at 1.5D lateral spacing. Data from Stallard et al. [39] and various Delft3D model definitions with varying grid sizes. The highlighted area in blue indicates deviation of 15% of the data from Stallard et al. [39]. Note the differences in scale.

Figure 56 shows the vertical velocity profile and stream wise turbulence intensity profile of the centre rotor, with data of Stallard et al. [39] and various Delft3D model definitions with varying grid sizes. The model definitions show the same influence of the discretization to the vertical velocity and stream wise turbulence intensity profile as the horizontal variation, except for Delft3D_3rotor_Hsubgrid. In Delft3D_3rotor_Hsubgrid the rotor is only sub-grid in the horizontal directions, the model definition does simulate the shape of the vertical velocity deficit profile.

The differences between the non-sub-grid models are small. At approximately x/D ≈ 7 downstream of the rotor the velocity of all the model definitions correspond to the experimental data within the desired accuracy. However, the sub-grid models do not resolve the shape of the velocity profile.
Numerical modelling

Figure 56 Vertical variation of A) velocity and B) stream wise turbulence intensity downstream middle rotor of a row of three rotors, with data of Stallard et al. [39] and various Delft3D model definitions with varying grid sizes. Note the differences in scale.

4.5.2 Horizontal large eddy simulation (HLES)

<table>
<thead>
<tr>
<th>Model definition</th>
<th>Mesh size ((\Delta x, \Delta y, \Delta z)) normalized by the rotor diameter (D=0.27) m.</th>
<th>HLES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delft3D_3rotor_coarse2</td>
<td>(0.185, 0.185, 0.056)</td>
<td>No HLES</td>
</tr>
<tr>
<td>Delft3D_3rotor_coarse4</td>
<td>(0.370, 0.370, 0.056)</td>
<td>No HLES</td>
</tr>
<tr>
<td>Delft3D_3rotor_Hsubgrid</td>
<td>(3.704, 3.704, 0.056)</td>
<td>No HLES</td>
</tr>
<tr>
<td>Delft3D_3rotor_subgrid</td>
<td>(3.704, 3.704, 1)</td>
<td>No HLES</td>
</tr>
<tr>
<td>Delft3D_3rotor_coarse2_HLES</td>
<td>(0.185, 0.185, 0.056)</td>
<td>HLES</td>
</tr>
<tr>
<td>Delft3D_3rotor_coarse4_HLES</td>
<td>(0.370, 0.370, 0.056)</td>
<td>HLES</td>
</tr>
<tr>
<td>Delft3D_3rotor_Hsubgrid_HLES</td>
<td>(3.704, 3.704, 0.056)</td>
<td>HLES</td>
</tr>
<tr>
<td>Delft3D_3rotor_subgrid_HLES</td>
<td>(3.704, 3.704, 1)</td>
<td>HLES</td>
</tr>
</tbody>
</table>

Table 4 Model definitions, their corresponding mesh size and use of HLES.
The use of HLES can possibly lead to better results and its effectiveness depends among others things on the mesh size. Therefore, the influence of HLES is checked for various mesh sizes. Model definitions with HLES and without are compared to data of Stallard et al. [39] in this section. The model definitions, their corresponding mesh size and usage of HLES can be found in Table 3. It is important to note that the influence of HLES in this section is checked in combination with the Z-layer model and steady flow. This may influence the effectiveness of the HLES. The influence of HLES in combination with the sigma-layer model can be found in Appendix K.

Figure 57 shows the longitudinal velocity and stream wise turbulence intensity on the centre line of single rotor with data of Stallard et al. [39] and model definitions which vary in use of HLES and mesh size. Note that the dashed lines may not be visible due to the same position of the uninterrupted lines. The influence of HLES on the centre line velocity is minimal. The non-sub-grid model definitions with HLES (Delft3d_1rotor_coarse2_HLES and Delft3d_1rotor_coarse4_HLES) shows that HLES is capable of producing stream wise turbulence intensities which fit better to the experimental data.

![Diagram showing velocity and turbulence intensity](image-url)

Figure 57 Velocity and stream wise turbulence intensity downstream of the axis of a single rotor with data of Stallard et al. [39] and model definitions which vary in use of HLES and mesh size. Note that in the top picture the dashed lines may not be visible due to the same position of the uninterrupted lines. In the bottom picture the black and cyan dashed lines are not visible for the same reason.
Numerical modelling

Figure 58 shows the lateral velocity profile and stream wise turbulence intensity at sections downstream of a row of three rotors, with data of Stallard et al. [39] and model definitions which vary in use of HLES and mesh size. The lateral velocity profile is practically unaffected by the HLES model, the dashed lines are not visible everywhere due to the same position of the uninterrupted lines. The stream wise turbulence intensities of the non-sub-grid models (Delft3d_3rotor_coarse2_HLES and Delft3d_3rotor_coarse4_HLES) have increased, which corresponds better to the turbulence intensities found in the wake of HATTs in the near wake. In the far wake these models overestimate the turbulent intensities. The sub-grid models do not produce high lateral shear, see section 4.5.1. and thus the HLES is not very effective for these models. The vertical velocity profile and stream wise turbulence intensities show the same features as the horizontal velocity profile and stream wise turbulence intensities, see Appendix J.

![Figure 58: Lateral variation of A) velocity and B) stream wise turbulence intensity downstream of a row of three rotors at 1.5D lateral spacing. Data from Stallard et al. [39] and model definitions which vary in use of HLES and mesh size. The highlighted area in blue indicates deviation of 15% of the data from Stallard et al. [39].](image)

The influence of HLES in combination with Z-layer model and steady flow on the velocity is negligible. The use of HLES for non-sub-grid model definitions results in a better representation of the turbulent intensities in the near wake. In the far wake the turbulence intensities are overestimated. Model runs in which the HATT rotor is sub-grid are hardly affected by the use of HLES.
4.5.3 Hydrostatic mode

This section discusses the influence of hydrostatic mode. The use of hydrostatic mode enables the selection of two additional model parameters:

- Grid layer model.
- Depth averaged flow.

The influence of these parameters on the simulation of the HATT is investigated here as well. Firstly, it is checked if the hydrostatic assumption is invalid at the inflow and near wake of the HATT, as mentioned in Paragraph 4.3. Therefore, the effect of hydrostatic mode is checked for a relative fine mesh. Sub-grid models do not simulate the inflow and near wake very accurately (see section 4.5.1), hence the effect of hydrostatic mode on sub-grid models is checked. Thirdly, the influence of the definition of the grid layer model is investigated. Finally, the influence of the depth averaged flow is investigated.

Model definitions with and without hydrostatic mode are compared to data of Stallard et al. [39]. See Table 5 for the model definitions and their corresponding mesh size and use of hydrostatic mode.

<table>
<thead>
<tr>
<th>Model definition</th>
<th>Mesh size (Δx, Δy, Δz) normalized by the rotor diameter D (D=0.27 m.)</th>
<th>hydrostatic mode or non-hydrostatic mode</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delft3D_1rotor</td>
<td>(0.093, 0.093, 0.056)</td>
<td>Non-Hydrostatic mode</td>
</tr>
<tr>
<td>Delft3D_1rotor_hydrostatic</td>
<td>(0.093, 0.093, 0.056)</td>
<td>Hydrostatic mode</td>
</tr>
<tr>
<td>Delft3D_1rotor_Hsubgrid</td>
<td>(3.704, 3.704, 0.056)</td>
<td>Non-hydrostatic mode</td>
</tr>
<tr>
<td>Delft3D_1rotor_Hsubgrid_hydrostatic</td>
<td>(3.704, 3.704, 0.056)</td>
<td>Hydrostatic mode</td>
</tr>
<tr>
<td>Delft3D_1rotor_subgrid</td>
<td>(3.704, 3.704, 1)</td>
<td>Non-hydrostatic mode</td>
</tr>
<tr>
<td>Delft3D_1rotor_subgrid_hydrostatic</td>
<td>(3.704, 3.704, 1)</td>
<td>Hydrostatic mode</td>
</tr>
</tbody>
</table>

Table 5 Model definitions, their corresponding mesh size and use of HLES.

Figure 59 shows the velocity and stream wise turbulence intensity downstream of the axis of single rotor, with data of Stallard et al. [39], Delft3D_1rotor and Delft3D_1rotor_hydrostatic. At first sight, the hydrostatic simulation seems to fit better to the data of Stallard et al. [39]. At the inflow the velocity gradient is steeper and consequently the minimum velocity is lower. Downstream of the rotor the vertical accelerations can be neglected [41]. Consequently, the rate of velocity recovery of the hydrostatic simulation is similar to the rate of velocity recovery of the non-hydrostatic simulation. The non-hydrostatic is more accurate in the near wake, under the condition that the minimum velocity is simulated well. Section 4.4.3 explains the difference in simulated minimum velocity and measured minimum velocity.
Numerical modelling

Figure 59 Velocity and stream wise turbulence intensity downstream of the axis of single rotor. With data of Stallard et al. [39], Delft3D_1rotor and Delft3D_1rotor_hydrostatic.

The lateral and vertical velocity profile and stream wise turbulence intensities of the hydrostatic model definition show no extraordinary features, see Appendix J.

A sub-grid model does not model the minimum velocity very well, see 4.5.1. Therefore, it is investigated what the influence of the hydrostatic mode is on sub-grid models.

Figure 60 shows the velocity and stream wise turbulence intensity downstream of the axis of a single rotor of sub-grid models which vary in usage of hydrostatic mode. The model definitions run in hydrostatic mode behave similar to those run non-hydrostatic model. The effect of hydrostatic mode on the velocity profile is small, see Appendix J.

The effect of hydrostatic mode in combination with sub-grid model is negligible. When sub-grid model definitions are used, it is advised to use hydrostatic mode, as it doesn’t significantly influence the accuracy.
Numerical modelling

![Graph of velocity and streamwise turbulence intensity downstream of the axis of a single rotor](image)

**Figure 60** Velocity and streamwise turbulence intensity downstream of the axis of a single rotor. With data of Stallard et al. [39], Delft3D_1rotor_Hsubgrid, Delft3D_1rotor_Hsubgrid_hydrostatic, Delft3D_1rotor_subgrid and Delft3D_1rotor_subgrid_hydrostatic.

**Grid layer model**
The influence of the grid layer model is investigated in this section. Model definitions with Z-layer and sigma-layer model are compared to data of Stallard et al. [39]. See Table 6 for the model definitions and their corresponding mesh size and use of grid layer model.

<table>
<thead>
<tr>
<th>Model definition</th>
<th>Mesh size ($\Delta x, \Delta y, \Delta z$) normalized by the rotor diameter D (D=0.27 m.)</th>
<th>Sigma or Z layer model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delft3D_1rotor_Hsubgrid_hydrostatic</td>
<td>(3.704, 3.704, 0.056)</td>
<td>Z-layer model</td>
</tr>
<tr>
<td>Delft3D_1rotor_Hsubgrid_sigma</td>
<td>(3.704, 3.704, 0.056)</td>
<td>Sigma-layer model</td>
</tr>
<tr>
<td>Delft3D_1rotor_subgrid_hydrostatic</td>
<td>(3.704, 3.704, 1)</td>
<td>Z-layer model</td>
</tr>
<tr>
<td>Delft3D_1rotor_subgrid_sigma</td>
<td>(3.704, 3.704, 1)</td>
<td>Sigma-layer model</td>
</tr>
</tbody>
</table>

Table 6 Model definitions, their corresponding mesh size and grid layer model.
Numerical modelling

Figure 61 shows that the use of the sigma layer model does not contribute to significant altered flow velocities. The vertical variation of the velocity (Figure 62) shows that there are discrepancies between the Z-layer and sigma-layer model at the bottom. The discrepancies are relatively small. The horizontal variations of the velocity and stream wise turbulence can be found in Appendix J.

The use of the sigma-layer model doesn’t contribute to large changes in sub-grid models. Note that the sigma-layer model is only tested in combination with a horizontal bottom.

![Graph](image)

Figure 61 Velocity and stream wise turbulence intensity downstream of the axis of a single rotor. With data of Stallard et al. [39], Delft3D_1rotor_Hsubgrid_hydrostatic, Delft3D_1rotor_Hsubgrid_sigma, Delft3D_1rotor_subgrid_hydrostatic and Delft3D_1rotor_subgrid_sigma.
Numerical modelling

![Figure 62](image)

**Figure 62** Vertical variation of A) velocity and B) stream wise turbulence intensity downstream of the middle rotor of a row of three rotors, with data of Stallard et al. [39], Delft3D_3rotor_Hsubgrid_hydrostatic, Delft3D_3rotor_Hsubgrid_sigma, Delft3D_3rotor_subgrid_hydrostatic and Delft3D_3rotor_subgrid_sigma. Note the differences in scale.

**Depth averaged flow**

The effect of the depth averaged flow is investigated in this section. Model definitions with multiple and a single vertical layer are compared to data of Stallard et al. [39]. Additionally, it is checked if the use of HLES can improve the results of a depth averaged simulation. See Table 7 for the model definitions and their corresponding mesh size and use of HLES. Note that the k-ε turbulence model cannot be applied to depth averaged models.

<table>
<thead>
<tr>
<th>Model definition</th>
<th>Grid size ((\Delta x, \Delta y, \Delta z)) normalized by the rotor diameter (D (D=0.27 \text{ m})).</th>
<th>HLES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delft3D_1rotor_Hsubgrid_hydrostatic</td>
<td>(3.704, 3.704, 0.056)</td>
<td>No HLES</td>
</tr>
<tr>
<td>Delft3D_1rotor_subgrid_hydrostatic</td>
<td>(3.704, 3.704, 1)</td>
<td>No HLES</td>
</tr>
<tr>
<td>Delft3D_1rotor_depth</td>
<td>(3.704, 3.704, 1.667)</td>
<td>No HLES</td>
</tr>
<tr>
<td>Delft3D_1rotor_depth_HLES</td>
<td>(3.704, 3.704, 1.667)</td>
<td>HLES</td>
</tr>
</tbody>
</table>

Table 7 Model definitions, their corresponding mesh size and grid layer model.
Numerical modelling

Figure 63 shows velocity downstream of the axis of a single rotor, with data of Stallard et al. [39] and model definitions which vary in vertical mesh size. Depth averaged flow models are within desired accuracy from approximately $x/D = 7$ downstream of the rotors. Depth averaged flow does not resolve the vertical velocity profile at all. Consequently, if the bottom is non-uniform there could possibly be large deviations of the resolved velocities. Depth averaged flow should be used with caution. If the vertical velocity profile is not of interest, the combination of a uniform bottom and depth averaged flow resolves the flow velocities from approximately $x/D = 7$ downstream of the rotor.

![Figure 63 Velocity downstream of the axis of a single rotor](image)

Figure 63 Velocity downstream of the axis of a single rotor. Data of Stallard et al. [39], Delft3D_1rotor_Hsubgrid_hydrostatic, Delft3D_1rotor_subgrid_hydrostatic, Delft3D_1rotor_Depth and Delft3D_1rotor_Depth_HLES.

4.6 Conclusion concerning modelling of HATTs in Delft3D

The HATT is modelled as a momentum loss term in Delft3D. The implementation method accounts for the absolute momentum extraction from the flow. However, it does not account for the non-uniform distribution of the momentum extraction. The implementation method ensures that the correct\textsuperscript{25} amount of momentum is extracted from the flow and that the correct\textsuperscript{26} power production of the HATT is calculated.

The validation shows that, if the influence of the support structure is taken into account, Delft3D is able to simulate the entire velocity field within the desired accuracy from approximately $x/D = 5$ downstream of the rotors. The support structure of actual HATTs should not be neglected. The velocity in between the rotors is simulated within the desired accuracy for distances smaller than $x/D = 5$. Delft3D is able to predict the shape of the merged wake well.

The inequalities of Delft3D compared to the experimental data in the near wake can be related to negligence of the non-uniform distribution of the momentum loss, differences between momentum extraction by a rotor and by a momentum sink term, due to the support structure and due to the influence of the discretization.

\textsuperscript{25} According to Equation 2.7.

\textsuperscript{26} According to Equation 2.3.
Numerical modelling

If the influence of the support structure is taken into account, the Delft3D model performs better than the theoretical models. The self-similar solution only reaches the desired accuracy from $x/D > 7$ downstream of the rotor and the shape of the velocity profile is predicted better by Delft3D compared to the theoretical models.

The accuracy of Delft3D is largely affected by the mesh resolution. If the rotor diameter is larger than the mesh size (sub-grid model), the near wake is modelled very inaccurately. The desired accuracy is only reached from approximately $x/D = 7$ downstream of the rotor and the shape of the velocity profile is not modelled accurately. The accuracy is less affected for non-sub-grid models, the differences between the non-sub-grid models is small. Hydrostatic mode in combination with non-sub-grid models shows large influences on the wake velocities. However, the sub-grid models are insensitive to the use of hydrostatic mode. Depth averaged flow should be used with caution as the vertical velocity profile shape is not simulated at all. The influence of HLES on the velocities is negligible and the effect of the grid layer model on the sub-grid models is relatively small. Note that the influence of the grid layer model is only tested for a uniform bottom.

This Chapter only focuses on modelling of the wake flow. It is important to note that the accuracy to which the ambient flow can be resolved also influences the accuracy of the wake flow. The validation cases consist of relatively simple cases in which the ambient flow velocity is modelled to relative high accuracies. Consequently, the realistic accuracy of the Delft3D model may be lower due to uncertainties of the ambient flow.

Overall Delft3D shows good potential to be used as a practical tool for modelling of HATTs. However, in the validation only tidal fences and not full tidal farms have been considered. The next Chapter consists of a case study, which illustrates how the validation of this Chapter contributes to the hydrodynamic and power production assessment of a tidal farm based on Delft3D.
The goal of the case study is to demonstrate the value of Delft3D to the assessment of the hydrodynamic impact and power production of a tidal farm. The case study illustrates the issues one encounters when modelling a tidal farm in the field. Firstly, the area of the case study is introduced. Secondly, attention is paid to design of the tidal farm. The validation cases of Chapter 4 differ from the situation of the case study. Therefore, the model definition used to model the tidal farm is discussed. The results of the simulation are presented and finally the conclusion is given.

5.1 Introduction

One of the criteria for a tidal farm is the presence of peak tidal currents in excess of 1 m/s [5]. Peak tidal current velocities of up to 1.8 m/s may be expected at the Marsdiep [56]. Consequently, the Marsdiep (see Figure 64) may be a possible location for the implementation of a tidal farm in the Netherlands. The Marsdiep is a tidal inlet of the Wadden Sea. The Wadden Sea is confined by the Wadden islands and the North Sea on one side and the mainland of the Netherlands, Germany and Denmark on the other side. The Wadden Sea is characterized by tidal flats, which run dry during low water and a network of tidal channels. It is an important ecological area and is used for fishery and recreation. Altering the inlet can have an effect on the water quality and water levels of a large part of the Wadden Sea. In short, a hydrodynamic impact assessment of a possible tidal farm in Marsdiep is important.

The tide in the Wadden Sea is dominated by the semi-diurnal lunar tide with a period of 12h and 25 min. Figure 65 shows the water level measured at Den Helder (see Figure 64).
5.2 Tidal farm Marsdiep

The most important aspect of the tidal farm is its power production. Ideally, the farm should be placed at a location which shows the highest potential for power production. To assess the potential of the power production of a location, the power density PD is used, see Equation 5.1.

\[ PD = \frac{1}{2} \rho U^3 \quad [W/m^2] \]

Equation 5.1

\( \rho \) and \( U \) being the density of water and depth averaged velocity respectively. Figure 66A shows the power density averaged over a (average) tidal period as derived from the model of Elias [56]. Based on the Figure, the most favourable location is the area highlighted by the black box. This area is located in the middle of the shipping lane and has an area of approximately 600 m by 1800 m. Lane separations are not uncommon in shipping lanes. If possible, an obstruction in the middle of the shipping lane is to be avoided. However, in this case there is a conflict of interest between clear and safe shipping lane and maximum energy production of the tidal farm. The minimal width for the normative ship\(^27\) is 450 m for a two way channel according to the PIANC guidelines for port approach channels. The widths at the Northern and Southern side of the boxed area indicated in Figure 66A are approximately 540 m and 500 m respectively. In short, there is enough space for the ships to pass the boxed area. The case study uses this coarse approximation to substantiate the location of the tidal farm in the middle of the shipping lane.

\(^{27}\) The normative ship is a landing platform dock of the Dutch Royal navy the Zr. Ms. Johan de Witt. The width of the ship is 29 m and the draft is 6 m.
Marsdiep case study

![Map of Marsdiep case study]

Figure 66 A) Power density averaged over an average tidal period [56]. B) Maximum velocity at spring tide [56]. C) bathymetry data [56]. D) location of the HATTs indicated with the markers. Waterway indicated by white dashed line and the location of the tidal farm is highlighted by the black box.

It is assumed that the tidal farm will consist of HATTs and they are selected on the base of the rated velocity and available water depth. The rated velocity is the velocity of the flow in which the turbine produces maximum power. In this case the maximum tidal current velocity is selected as the rated velocity. The maximum depth averaged tidal velocity at Den Helder is about 2 m/s [56]. The minimum water depth at the designated area is about 25 m. The HATTs are placed as high as possible in the water.

77
Marsdiep case study

column to optimize power production. However, at the free-surface a minimal depth of 6 m below LAT (lowest astronomical tide) is reserved for the possibility that a ship accidentally enters the tidal farm. Consequently, there is 17 m water depth left for the HATT. The Tocardo T500 has a rated velocity of 2 m/s, a rotor diameter of 14.2 m and a rated power production of 232 kW. The Tocardo T500 is the selected HATT in this case study. The power coefficient can be calculated by evaluating the power production at the rated velocity. The thrust coefficient is assumed to be 0.8 based on realistic values [39, 48, 60, 61] and is assumed to be constant\textsuperscript{28} [62]. See Table 8 and Figure 67 for the characteristics of the Tocardo T500.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor Diameter [m]</td>
<td>14.2</td>
</tr>
<tr>
<td>Swept area [m\textsuperscript{2}]</td>
<td>158</td>
</tr>
<tr>
<td>Number of blades [-]</td>
<td>2</td>
</tr>
<tr>
<td>Rated velocity [m/s]</td>
<td>2.0</td>
</tr>
<tr>
<td>Rated power [kW]</td>
<td>232</td>
</tr>
<tr>
<td>Cut in speed [m/s]</td>
<td>0.4</td>
</tr>
<tr>
<td>Cut out speed [m/s]</td>
<td>2.6 m/s</td>
</tr>
</tbody>
</table>

Table 8 Specification of the Tocardo T500.

The tidal farm contains 123 HATTs spaced by a distance of 2 rotor diameter from rotor centre point to rotor centre point. The stream wise distance between the rows of HATTs is 20 rotor diameters, see Figure 66D. The most Eastern row of HATTs has three HATTs less compared to the other rows due to insufficient water depth, see Figure 66D. The farm has a rated power of 28.5 MW. The array formation has been chosen to maximize the number of HATTs in the area while taking into account the negative influence the wake of an upstream HATT can have on the power production of a HATT placed downstream. Studies such as Bai et al. [63] show that staggered array formations can be more effective

\textsuperscript{28} Below the rated velocity the power and thrust coefficient can be approximated to be constant [62].

\textsuperscript{29} Values of power and thrust coefficient assumed. Therefore, the figure does not show the exact characteristics of the Tocardo T500.
than rectilinear array formations. However, the conclusion of Bai et al. [63] is based on a case with straight steady flow. The flow at the tidal farm of the Marsdiep varies in time and can have an oblique flow with the HATTs. Consequently, it is more difficult to design a staggered array formation for the Marsdiep tidal farm which is more effective than a rectilinear array formation. Therefore, as a first estimate the array formation as shown in Figure 66D is used. Due to the relatively large tidal currents it is assumed that the HATTs are bottom mounted.

Note that the case study is based on crude assumptions. In reality the design of a tidal farm requires more fine tuning, for example the criteria for the location of a tidal farm depends on many aspects such as: water depth, tidal characteristics, shipping lanes, ice formation, storm climate, morphological effects, stratification, bottom material, ecological value, wave climate, etc.

5.3 Expectations of the power production

Based on the selected HATT of the tidal farm one can make estimations of the power production of the tidal farm.

Elias [56] has set up, calibrated and validated a Delft3D-FLOW model in order to investigate the Marsdiep inlet. The depth averaged flow velocities calculated from this model will be used to estimate the power production of the tidal farm.

The power production of one HATT is estimated by using Equation 2.3 and the velocities resolved by the model of Elias [56]. Note that the HATT is not simulated in this model, only an estimate based on the flow velocities is made.

\[ P = \frac{1}{2} \cdot \rho \cdot \int \cdot C_p \cdot u_0^3 \cdot dA \rightarrow P = \frac{1}{2} \cdot \rho \cdot C_p \cdot A \cdot u_0^3 \]  

Equation 2.3

The monthly average power production of the HATT based on the flow velocities in the month of January 1999 is estimated to about 42.9 kW which comes down to 18.5 % of the rated power. The relative monthly averaged power production can likely be increased by choosing a HATT with a lower rated velocity.

5.4 The model definition

The model definition used to investigate the hydrodynamic impact of the tidal farm is described here. Firstly, the requirements of the model definition to assess the hydrodynamic impact of the farm are discussed and secondly attention is paid to the ambient turbulence.

5.4.1 Model settings

In this case study the model of Elias [56] has been modified to investigate the tidal farm. The full model definition details can be found in Elias [56]. This section elaborates on the relevant model settings of this modified model definition.

---

30 More effective averaged in time.
Marsdiep case study

In order to accurately predict the power production of the farm, it is necessary to resolve the velocity of the wake within 20 rotor diameters downstream of the HATTs, as the HATTs are positioned 20 rotor diameters downstream of each other. The validation cases of Chapter 4 are used to set the criteria for the model definition of the case study. However, the case study differs from the validation cases on the following points:

- The (bottom) geometry: All the conclusions of Chapter 4 are based for the case with a flat bottom. The bottom in the case study is not flat, see Figure 66C. The (bottom) geometry has an influence on the (vertical) mesh size and grid layer model.
- The HATTs are positioned downstream of each other in the case study. In Chapter 4 only tidal fences are considered. Due to inaccuracies of the resolved inflow of the HATTs positioned downstream of other HATTs, the accuracy of the power production of the total tidal farm is lower than the accuracy found in Chapter 4.
- The current velocity in the case study is time dependent. Appendix C concludes that the steady state assumption is valid for the wake of HATTs. Thus, the differences due to the time dependency of the tidal current are expected to be small.
- In the case study oblique angles between the current and turbine are possible. Only the reference velocity component normal to the turbine is taken into account in the implementation of the HATT, see Appendix E. Consequently, the actual trust force and power production of the turbine can differ due to this non-zero angle.
- In the case study no turbulent boundary conditions have been prescribed. Section 5.3.2 elaborates on the ambient turbulence in the case study.

Despite the differences, the following model definitions settings have been set, based on the findings of Chapter 4:

- Mesh size: According to Chapter 4 it is permitted to have a sub-grid HATT in order to obtain the desired accuracy at 20D downstream. The model definition of Elias [56] has a horizontal mesh size of approximately 100 m at the location of the tidal farm. In order to avoid that multiple HATTs are modelled in one grid cell the horizontal mesh has been refined to about 15 m at the tidal farm. Due to the non-uniform bottom of the case study (see Figure 66C), 10 vertical mesh layers have been selected. The model of Elias [56] has been modified from a depth average to a model with 10 vertical mesh layers.
- Hydrostatic mode: non-hydrostatic effects only play a role in close vicinity of the HATT. In the far wake non-hydrostatic effects are negligible. Thus, the model is run in hydrostatic mode.
- Sigma layer model: The differences between Sigma-layer model and Z-layer model on the wake of the HATT is negligible in the far wake for a flat bottom. Even though the case study does not have a flat bottom, the sigma layer model is selected.
- For the three-dimensional turbulence the k-ε model has been used. Chapter 4 showed that the influence of HLES (in combination with Z-layer model and steady flow) is negligible on the wakes of the HATTs. Thus, HLES is not selected. The horizontal background viscosity has been set to 1 m²/s.
The thrust and power coefficient of the HATT have been calibrated to the mesh size, as described in Chapter 4. Consequently, the reference velocity of the HATTs is set at the turbine, e.g. $u_{\text{ref}} = u_d$. See Appendix E for details.

As a result of the model settings, the model definition of the case study corresponds best to model definition Delf3D_3rotor_Hsubgrid_hydrostatic of Chapter 4. Note that the support structure of the HATT is not modelled. Thus, the contribution of the support structure on the wake is not taken into account in this case study.

Note that in this case study the farm is not optimized based on the results of the Delft3D model. Optimization based on the Delft3D model would require the model to resolve the velocity field inside the farm in more detail. A 3D mesh resolution which is in the order of approximately 0.37D in combination with hydrostatic mode and the k-ε turbulence model should give sufficient detail of the velocity field inside the farm, see Chapter 4.

5.4.2 Ambient turbulence
Because the turbulence is important for the recovery of the wake, it is checked if the model definition shows realistic turbulence intensities.

Estimated from data of Osalusi [47] typical time averaged maximum turbulence intensities found at HATT implementations areas in the field are between 6% and 18%. The time averaged maximum turbulence intensity found in the area of the tidal farm in the model is 24%, see Figure 68. Appendix F elaborates how to obtain the turbulence intensity from the turbulent kinetic energy. The time averaged turbulence intensity of the model is higher than desired. However, its value does not deviate severely from the data of Osalusi [47].

![Figure 68 Turbulence intensity of the bottom layer averaged over a tidal period in %. Location of the tidal farm is indicated by the black box.](image)

---

31 Averaged over a tidal period.
32 The turbulent kinetic energy is an output of Delft3D-FLOW.
5.5 Results
The goal of the case study is to illustrate the capability of Delft3D to assess the hydrodynamic impact and power production of a tidal farm. The results show the hydrodynamic impact and power production of the farm in the Marsdiep area.

5.5.1 Hydrodynamic impact
Firstly, it is checked if the model behaves as one may expect from Chapter 4 and secondly two scenarios are compared to each other:

- Scenario 0: Reference condition with no changes to the Marsdiep area.
- Scenario 1: Tidal farm present in the Marsdiep area.

Check on the model
Figure 69 shows the maximum velocity during spring tide at mid depth. Figure 69A shows that the farm forms a significant blockage to the flow. A wake behind the tidal farm can be identified and the largest flow velocities are found adjacent to the farm. Figure 69B shows that the flow in between the HATTs increases. Figure 69C shows the increased velocity upstream of the HATT and the recovery of the wake. It can clearly be seen that the inflow velocity of the HATT on the downstream row is affected by the upstream HATT. Similar to Delf3D_3rotor_Hsubgrid_hydrostatic the minimum velocity on the centre line can be found around x/D = 2. Due to the higher ambient turbulence and lower thrust coefficient of the HATT it is expected that the velocity deficit at x/D = 16 of the case study is lower compared to the experimental data. The velocity deficit at x/D = 16 of the case study is approximately 0.02 whereas the velocity deficit of the experimental data of three rotors spaced 1.5D is about 0.19, see Chapter 3. Thus, the velocity deficit matches the expectations.

Figure 70 shows the vertical variation of the velocity downstream of the HATT indicated in Figure 69. Due to blockage of the flow, the flow velocity above and below the HATT increases. As the wake develops, it can clearly be seen that the wake recovers and the velocity distribution approaches a logarithmic velocity profile. In accordance to the experimental data of Chapter 3, the minimum velocity in the wake can be found below the centre line of the wake, this illustrates the influence of the sea bottom. It is not clear if the wake is bounded in the vertical direction, as it seems that from x/D = 11.31 the undisturbed velocity has recovered. However, Figure 69C indicates that the velocity at x/D = 11.31 is lower than the velocity found upstream of the HATT. Thus at x/D = 11.31, the wake has influenced the velocity of the entire water depth. In the experimental data and Delf3D_3rotor_Hsubgrid, which have a lower blockage factor and lower thrust coefficient, the wake is bounded by the vertical direction at approximately x/D = 8 a 10. Note that at x/D = 20 there is another HATT positioned.

The model of the case study behaves as one may expect from the experimental data and findings of Chapter 4.
Marsdiep case study

Figure 69 A) Maximum velocity during spring tide (21 January 19:00) in the middle of the water column. HATTs indicated with markers. B) Detailed view of the maximum velocity in the middle of the water column during spring tide. HATTs indicated with markers. C) Streamwise velocity in the middle of the water column up- and downstream of the HATT indicated by the black marker in A) and B). The HATT is located at x/D = 0.

Figure 70 Vertical variation of the velocity downstream of the HATT indicated in Figure 69. Rotor centre located at water depth of -14.35 m.
Marsdiep case study

**Comparison to the reference scenario**
The difference between scenario 0 and scenario 1 are presented in this section.

Figure 71 shows the velocity difference in the middle of the water column at spring high water level. The largest differences between the scenarios are found during spring high and spring low water. A relative large area is affected by the tidal farm. Locally, downstream of the farm velocity reduces in the order of 0.6 m/s (33% of maximum velocity of scenario 0). Directly adjacent to the farm the velocity increases by about 0.4 m/s (22% of maximum velocity of scenario 0). The tidal farm has a significant influence on the flow velocities. The velocity difference during spring low water level displays similar features as the difference during spring high water level, see Appendix J. The change of velocity due to the tidal farm reflects on the power density.

Figure 72 shows the difference in power density averaged over a tidal period. The tidal farm extracts power and redistributes the power density. The tidal farm can create differences in the power density up to 33%.

![Figure 71](image1.png)

**Figure 71** Velocity difference in m/s in the middle of the water column at spring high water level. Values between -0.01 and 0.01 m/s are not shown.

![Figure 73](image2.png)

**Figure 73** shows the water level difference during high water spring tide. Similarly to the change of velocity, the water level difference shows that a large area is affected by the farm. The maximum induced change is small about 3.5% but not negligible. The difference in water level during spring low water level shows resembling features to the difference during spring high water level, see Appendix J.
The mixing time scale $T$ is defined here as the average volume of the basin $V$ divided by the average net discharge $Q$, see Equation 5.2.

$$ T = \frac{V}{Q} $$

Equation 5.2

Ridderinkhof et al. [64] state that the average volume of the Marsdiep tidal basin is $2890 \times 10^6$ m$^3$. Thus, the mixing time scale of scenario 0 and scenario 1 can be estimated by calculating the net average discharge through the Marsdiep inlet from the Delft3D model definitions. Note that the influence from river discharges and influence due to interactions with the adjacent tidal basin has been neglected. For a description of a more sophisticated method to estimate the mixing time scale see Ridderinkhof et al. [64].
Due to the tidal farm the net average discharge changes by 7.6% and consequently the mixing time scale of the Mardiep basin reduces by 7.0%. As a first estimate the mixing time scale of the tidal basin can be significantly affected by the tidal farm. Note that a significant change of the mixing time scale will induce morphological changes to the Mardiep basin. The tidal farm, can only be authorised if the morphological changes it induces are known. Hence, the morphological changes induced by HATTs should be investigated.

The tidal farm can have a significant effect on the current velocities and discharge at the Marsdiep inlet. The effects on the water level are small but not negligible.

5.5.2 **Power production**

Figure 74A shows the average daily power production of the tidal farm over a period of one month. The monthly average power production of approximately 2.6 MW is only 9.1% of the rated power. It is sufficient to supply about 6875 average Dutch households [65]. Recall that the relative monthly averaged power production of one turbine is estimated to 18.5%. Hence, the low value of the relative monthly power production of the farm indicates that further optimization of the array formation will likely lead to large gains. The variation of the power production corresponds to the spring neap cycle of the tide. Consequently, if the flow velocities are well predicted, the power production of the farm can be predicted relatively accurately as well. Figure 74B shows the instantaneous power production on the 8<sup>th</sup> of January. Notice that there are periods during the day that the tidal farm does not produce power. The variation of the power production during the day is important, as the demand for power during the day varies.

![Figure 74 Power production of the tidal farm. A) Shows the daily averaged power production of the farm in the month of January 1999. Monthly averaged power production of 2.6 MW is indicated by the magenta line. B) Shows the instantaneous power production of the farm on the 8<sup>th</sup> of January. Note that the 8<sup>th</sup> of January is highlighted in A).](image-url)
5.6 Conclusion
The case study consists of a first estimate of the design and assessment of the tidal farm. To substantiate the design, coarse approximations are used. However, the case study shows the capability of Delft3D to assess the hydrodynamic impact and power production of a tidal farm.

Based on findings of Chapter 4 the model definition used in the case study has been set up to simulate the velocity of the wake within the desired accuracy of 15% within 20 rotor diameters. Hence, the rotor diameter is smaller than the mesh size, hydrostatic mode is used and 10 mesh layers have been used. Optimization of the farm based on Delft3D-FLOW requires a 3D grid with a mesh size of approximately 0.37D in combination with hydrostatic mode and the k-ε turbulence model.

Important differences between the validation cases and the case study are:

- Differences between the (bottom) geometry
- Differences between array formation of the HATTs
- Differences of the ambient turbulence

Despite the differences, the model of the case study behaves as one may expect from the experimental data and findings of Chapter 4.

The influence of the tidal farm on the tidal current velocities and discharge is significant. A relatively large area is affected by the tidal farm. As a result, the potential power decreases, the power density is redistributed and the mixing time scale is affected. The influence of the farm on the water level extends to a large area and is not negligible.

The relative monthly averaged power production of one turbine is estimated to 18.5% of the rated power. Its moderate value indicates that the rated velocity of the HATT may be too high. The power production based on the Delft3D model shows that the array formation and the effect of the farm on the tides can significantly influence the power production, as the monthly average power production of approximately 2.6 MW is only 9.1% of the rated power. The variations of the tides are reflected on the power production. Tidal currents can often be well predicted and subsequently the power production of the farm can be predicted relatively accurately as well.
6 Discussion

The content of the report can roughly be divided in four parts: Theoretical knowledge, experimental data, numerical modelling and case study. The discussion is structured accordingly.

6.1 Theoretical knowledge
The hydrodynamics of HATTs shows similarities to wind-turbines, propellers and aerodynamics of planes and helicopters. Consequently, the theoretical knowledge concerning the hydrodynamic of HATTs is relatively well described. The most important difference of HATTs, compared to wind-turbines, is the proximity of the free-surface and sea bottom and adjacent turbines in closely spaced array formations of the farms. The proximity of the boundaries gives rise to most of the uncertainties found in the theoretical knowledge.

A discussion point not related to the influence of the free-surface and sea bottom is the self-similar behaviour of the wakes of HATTs. Koshnevis et al. [66] show that not all wake flows automatically display self-similar behaviour. The wake of a HATT exhibits self-similar behaviour, however one should keep in mind that this may not always be valid. Furthermore, it is not clear from which distance the self-similar solution can be applied to the wake of a HATT.

Note that not all the hydrodynamics concerning HATTs are included in the thesis, for example the effect of waves and oblique current angles are excluded. Exclusions disregarded, the theoretical knowledge is sufficient to describe the hydrodynamics of the HATTs relatively well.

6.2 Experimental data
Stallard et al. [39] simulate the wake of full scale HATTs by the use of scale experiments. Accordingly, the discussion of the experimental data elaborates on the conditions of these experiments to simulate full scale HATTs. In the thesis the data of Stallard et al. [39] has been used as validation data. Hence, the discussion also elaborates on the suitability of the experimental data as validation data.

To minimise scaling effects Fr scaling in combination with Re numbers in the turbulent regime are used. The rotor diameter of the HATT is 0.27 m and is situated in a 5 m wide and 12 m long flume with a water depth of 0.45 m. The mean velocity at z = 0 (the middle) is 0.47 m/s. ‘At 1:70th geometric scaling, these experiments represent a rotor diameter of 19 m, water depth of 31.5 m and applying Froude scaling a mean incident flow velocity of 3.93 m/s. These scaled dimensions shows realistic dimensions of full scale HATTs in the field.

The ambient turbulence intensities of the experiments of Stallard et al. [39] should be representative for values found in the field. Estimated on data of Osalusi [47] it is feasible to assume that the ambient turbulence intensity in the experiments of Stallard et al. [39] are representative for the field.
The experiments of Stallard et al. [39] assume a steady state whereas tidal currents vary in time. Appendix C concludes that the steady state assumption of Stallard et al. [39] is valid in the case of the experiments, as in the worst case scenario the change of the ambient current is about 4.4% in the range of interest.

The assumptions made in the experiments of Stallard et al. [39] seem reasonable to simulate the wake of a full scale tidal fence consisting of HATTs.

The experimental data contains errors and discrepancies, such as measurement errors, the asymmetric ambient flow, limited information concerning a single wake, limited information concerning turbulence intensities, etc. However, despite these errors the realistic dimensions of the HATT compared to the adjacent HATTs and water depth in combination with the available supplementary data make the experiments of Stallard et al. [39] suitable for validation of the numerical model.

6.3 Numerical model
In order to simulate the hydrodynamics of HATTs in Delft3D several assumptions have been made. The assumptions are discussed here.

The validation of the Delft3D is based on the experiments of Stallard et al. [39]. Any errors or discrepancies (see Paragraph 6.2) in the experiments influence the validation of Delft3D. The validation of Delft3D is strongly dependent on the experiments of Stallard et al. [39].

The support structure plays a significant role in the validation. Yet, its influence in the validation cases is overestimated.

The HATT is modelled as a momentum loss term in Delft3D. The momentum loss term models a porous disc which in turn models the rotor of the HATT. Consequently, discrepancies are to be expected in the near wake of the HATT and the near wake of the momentum sink, as implemented in Delft3D. Discrepancies such as the production of turbulence, differences between the swirl angles, differences between shed vortices etc. can be expected. Furthermore, the implementation method of the HATT in Delft3D does not account for the non-uniform distribution of the momentum loss. Thus, differences in the distribution of the momentum extraction between an actual HATT and the modelled HATT of Delft3D exist.

The k-ε model is used to model the turbulence. The k-ε shows discrepancies compared to actual turbulence characteristics. For example, it is known that the k-ε turbulent model dissipates the turbulent kinetic energy too fast [51]. The influence of the HLES model is only tested in combination with steady flow. Due to the implementation method of HLES in Delft3D it is not effective for steady flows without velocity fluctuations [49].

The largest concern of the Delft3D model regards the dependency of the validation on the experiments of Stallard et al. [39]. The data only consist of tidal fences and lacks information concerning cross stream turbulence. The crude representation of the support structure is likewise of concern.
6.4 **Case study**

This paragraph elaborates on the assumptions made in the case study in order to evaluate a tidal farm in the Marsdiep.

The case study differs from the validation cases. Important differences are the (bottom) geometry, array formations of actual tidal farms, time dependency of the direction and velocity of the tidal current and differences in the ambient turbulence. Not all the aspects, such as modelling of HATTs with a non-uniform bottom, of the case study have been validated. Nonetheless, the model of the case study behaves as one may expect.

The farm of the case study is only a conceptual design of the farm. Relevant assumptions regarding the design of the tidal farm are the exclusion of the effect of the stratification and morphology. It may be possible for either of these influences to play a dominant role in the design of the tidal farm. Another significant aspect, which has been neglected, is the effect of the waves on the farm. Note that the argument to place the tidal farm in the middle of the shipping lane needs more fine tuning in order to be justified. There are considerable more criteria which should be included in full case study such as storm climate, bottom material, influence of support structure, ecological value, marine life protection, costs, etc. However, as a first estimate for the design, the tidal farm shows several relevant aspects.
## Conclusion & recommendations

### 7.1 Conclusion

The main question of the thesis is: What are the important physical processes involved in the implementation of horizontal-axis tidal turbines (HATTs) and how can these be represented in a numerical model?

#### 7.1.1 Theoretical knowledge and observations

The momentum loss of the flow due to the power production and the wake induced by the momentum loss are important flow characteristics concerning the hydrodynamics of HATTs. The following conclusions can be made concerning the wake of HATTs:

- The wake can significantly influence the power production of HATTs in array formations. As at $x/D = 20$ ($x$ being the downstream directional coordinate and $D$ the diameter of the rotor) downstream of the HATT, the velocity deficit is in the order of 10% to 20%.
- The flow in the near wake ($x/D < 4$) is complex, highly unsteady and governed by detailed turbine characteristics.
- The thrust force on the HATT and the dimensions of the swept area are the only turbine parameters which influence the far wake ($x/D > 4$).
- The structure of the far wake is mainly determined by convection and turbulence mixing.
- The turbine generated turbulence plays a substantial role in the development of the near wake, however the ambient turbulence is more influential in the far wake.
- The influence of free-surface and bottom on the wake is significant. It causes the near wake to be non-axi-symmetric. The far wake may be confined by these boundaries, similarly as shallow wakes.
- Due to the close lateral spacing of HATTs, wake-wake interactions in the far wake lead to merging of wakes.
- The far wake of the HATT can be described by the self-similar solution.

In general, the theoretical knowledge is sufficient to describe the hydrodynamics of the HATTs relatively well.

#### 7.1.2 Numerical modelling

The rotor of the HATT is modelled by a momentum sink. The momentum sink models the extraction of momentum from the flow and the induced wake. The following conclusions can be made concerning the modelling of HATTs in Delft3D-FLOW:

- A ‘calibration’ procedure enables the model to transform the thrust and power coefficient (used to calculate the thrust force and power production), defined by the undisturbed velocity, to coefficients defined by the velocity through the HATT. These transformed coefficients can subsequently be used as input for the model of the tidal farm.
Conclusion & recommendations

- In combination with a mesh size in the order of 0.37D, non-hydrostatic mode, appropriate ambient turbulence and accounting for the influence of the support structure, Delft3D-FLOW is able to simulate the entire velocity field within the desired accuracy of 15% from approximately $x/D = 5$ downstream of the rotors.
- Delft3D-FLOW is able to predict the shape of the merged wake well and to a similar accuracy as a single wake.
- Delft3D-FLOW simulates the velocity more accurately compared to the theoretical models.
- The accuracy of Delft3D-FLOW is strongly affected by the mesh size. When the horizontal mesh size becomes larger than the rotor diameter, the near wake cannot be modelled. However the far wake is modelled within the desired accuracy from $x/D = 7$ downstream of the HATT. The vertical mesh shows a similar trend.
- The support structure of actual HATTs plays a significant role on the total wake. The validation indicates that the support structure may play a significant role in the near wake.

Overall Delft3D shows good potential to be used as a practical tool for the implementation of a tidal farm. In Paragraph 7.2 recommendations to improve Delft3D-FLOW are given.

7.1.3 Case study

The case study consists of an estimate of the power production and hydrodynamic impact of a conceptual tidal farm in the Marsdiep area.

- The implementation method of the HATT is suitable to realistic models of tidal farms, as the model definition of the case study behaves as one may expect from the experimental data and validation of Delft3D-FLOW
- A large area is affected by the tidal farm. The influence of the tidal farm on the tidal current velocities and discharge are significant and its influence on the water level is small, but not negligible.
- The farm has a small power production compared to the rated power. The monthly averaged power production of 2.6 MW is only 9.1% of the rated power.

The latter two points underline the importance of an impact assessment and further optimization of the tidal farm

7.2 Recommendations

Recommendations for future work based on the findings and assumptions used in the thesis are presented here.

7.2.1 Theoretical knowledge

The theoretical knowledge describes the hydrodynamics reasonably well. However, some reasonable improvements can be made by further investigation of the influence of the free-surface and sea bottom. Investigating how the influence of the bottom causes the minimum velocity in the wake to shift downward contributes to a better understanding of its influence and may lead to a more accurate
model of the wake. Investigating when and to what extend the wake of a HATT shows similarities to a shallow wake can improve the understanding of the large scale effects.

In the thesis some aspects such as the effect of waves, oblique current directions and morphological changes have been neglected. Including these effects lead to a more realistic representation of the impact of HATTs.

7.2.2 **Numerical modelling**

Recommendations concerning the numerical model are related to further validation an to improving the Delft3D-FLOW model.

The validation can be improved by:

- Implementation of a more accurate schematization of the support structure in the validation cases.
- Comparison of Delft3D model results to more experiments of Stallard et al. [39] or other experiments. Note that computational Fluid Dynamics (CFD) simulations could also be used as validation data.
- Checking the influence of HLES in combination with non-steady flow.
- Testing the influence of the Z-layers and sigma-coordinates for non-uniform bottoms.

Delft3D-FLOW can be improved by:

- In this study the distribution of the thrust coefficient is assumed constant over the area of the momentum sink. It is recommended to investigate whether a more realistic distribution leads to a better representation of the wake.
- It is not clear to what extend an empirical turbulence function for the turbine generated turbulence ascribed to the porous disc would improve the Delft3D-FLOW model results. This option should be investigated.
- Alternative to the modelling of the HATT as a porous disc, the HATT could be modelled on a method based on blade element theory. This theory describes the lift and drag forces on the blades by considering blade sections. It is not clear to what extend the use of this method to Delft3D-FLOW could contribute to its improvement. Batten et al. [27] conclude that this method corresponds better to experimental data compared to the method based on the porous disc, for a numerical model based on the Reynolds-averaged Navier-Stokes equation. Note that blade element theory requires detailed blade geometry information.
8 References

References


References

57. Google Earth.
Appendices

A. Actuator disc theory (Betz theory)
This Appendix gives the derivation of the actuator disc theory.

An actuator disc is a porous disc which is placed perpendicular to the undisturbed flow. It is assumed that the flow is incompressible, invicid and remote from boundaries. The pressure on the disc is assumed uniform and a steady state situation is considered. A control volume is introduced which satisfies the conservation of mass and does not allow for mass transport through the boundaries. Hence, due the pressure drop, the velocity decreases downstream of the disc, as a result the control volume must expand to satisfy the mass conservation see Figure 75 [11].

![Diagram of actuator disc and control volume](image)

Figure 75 control volume with actuator disc, flow is from left to right [11]

The following derivation is from Palm [11]. The conservation of mass demands the flow rate in every cross section to be equal:

\[ \rho \cdot A_0 \cdot u_0 = \rho \cdot A_d \cdot u_d = \rho \cdot A_w \cdot u_w \]  \hspace{1cm} \text{Equation A.1}

In which \( \rho \), \( A \) and \( u \) are density, cross sectional area and velocity respectively, where subscript 0 stands for the undisturbed flow, the subscript \( d \) stands for the conditions at the actuator disc and the subscript \( w \) stands for the conditions in the far wake where the pressure is similar to the free stream condition. The velocity at the disc is expressed using an axial flow induction factor \( a \):

\[ u_d = u_0(1 - a) \]  \hspace{1cm} \text{Equation A.2}

The force on the actuator disc is equal to the rate of momentum exchange:

\[ (p^+ - p^-) \cdot A_d = (u_0 - u_w) \cdot \rho \cdot A_d \cdot u_d \]  \hspace{1cm} \text{Equation A.3}
In which \( P^+ \) and \( P^- \) are the pressures upstream and downstream of the disc respectively. Note that Equation A.3 assumes that the pressure distribution on the disc is uniform. Generally such a distribution leads to an overestimation of the forces on the disc [67]. Substituting Equation A.2 in Equation A.3 gives:

\[
(P^+ - P^-) \cdot A_d = (u_0 - u_w) \cdot \rho \cdot A_d \cdot u_0 (1 - a)
\]  

Equation A.4

To evaluate the flow the Bernoulli equation is applied to the flow. The equation of Bernoulli for two theoretical cross sections 1 and 2 is given by:

\[
[P + \frac{1}{2} \rho \cdot u^2 + \rho \cdot g \cdot z]_1 = [P + \frac{1}{2} \rho \cdot u^2 + \rho \cdot g \cdot z]_2
\]  

Equation A.5

For simplicity the gravitational energy term \( \rho \cdot g \cdot z \) is neglected as no changes in heights are expected. Applying the Bernoulli equation to the system of the actuator disc gives:

For the upstream part

\[
P_0 + \frac{1}{2} \rho \cdot u_0^2 = P^+ + \frac{1}{2} \rho \cdot u_d^2
\]  

Equation A.6

For the downstream part

\[
P_0 + \frac{1}{2} \rho \cdot u_w^2 = P^- + \frac{1}{2} \rho \cdot u_d^2
\]  

Equation A.7

Subtracting Equation A.6 from Equation A.7 gives the pressure drop over the disc:

\[
P^+ - P^- = \frac{1}{2} \rho (u_0^2 - u_w^2)
\]  

Equation A.8

Combining Equation A.8 with force on the actuator disc Equation A.4 gives:

\[
\frac{1}{2} \rho (u_0^2 - u_w^2) \cdot A_d = (u_0 - u_w) \cdot \rho \cdot A_d \cdot u_0 (1 - a)
\]  

Equation A.9

Using relation Equation 2.10, Equation 2.9 can be rewritten, see Equation 2.11.

\[
(u_0 - u_w) = 2 \cdot u_d = 2 \cdot u_0 (1 - a)
\]  

Equation A.10

\[
u_w = u_0 (1 - 2a)
\]  

Equation A.11
This gives an expression for the velocity behind the disc. The force is often represented with the dimensionless parameter the trust coefficient $C_t$. The force on the disc is given by:

$$ F = \frac{1}{2} \cdot C_t \cdot \rho \cdot \int u_0^2 \cdot dA \rightarrow F = \frac{1}{2} \cdot C_t \cdot \rho \cdot A \cdot u_0^2 $$

Equation A.12

Equation A.12 assumes uniform velocity distribution over the disc. Note that the thrust coefficient is a function of the TSR similar as the power coefficient. If one substitutes the velocity relations of Equation A.2 and Equation A.11 in Equation A.12, after some algebra we get the following relation:

$$ F = 2a \cdot (1 - a) \cdot \rho \cdot A_d \cdot u_0^2 = \frac{1}{2} \cdot C_t \cdot \rho \cdot A \cdot u_0^2 $$

Equation A.13

This gives an expression for $C_t$:

$$ C_t = \frac{2a \cdot (1 - a) \cdot \rho \cdot A_d \cdot u_0^2}{\frac{1}{2} \cdot \rho \cdot A \cdot u_0^2} = 4a \cdot (1 - a) $$

Equation A.14

Some rearrangement of Equation A.14 gives an expression for $a$:

$$ a = -\left(\frac{1}{\sqrt{4 - 4 \cdot C_t^2}} - \frac{1}{2}\right) = \frac{1}{2} - \frac{1}{2} \cdot \sqrt{(1 - C_t)} $$

Equation A.15

If one substitute this expression (Equation A.15) of $a$ in the velocity relation $u_w = u_0(1 - 2a)$ (Equation A.11) one can eliminate $a$, which gives the velocity behind the disc as a function of the thrust coefficient $C_t$ [31].

$$ u_w = u_0 \cdot \sqrt{(1 - C_t)} $$

Equation A.16

The velocity at the disc ($u_d$) can now be expressed in term of $C_t$ with Equation A.2 and Equation A.15.

$$ u_d = u_0 \cdot \left(\frac{1}{2} + \frac{1}{2} \cdot \sqrt{(1 - C_t)}\right) $$

Equation A.17

The velocities have been expressed as a function of the thrust coefficient.

**Free-surface proximity model**

The free-surface proximity model describes the effect of the proximity of the boundaries (free surface and bottom) on the relations described in the actuator disc theory. The free surface model is applicable to a blocked configuration of HATTs such as a linear array [20]. Basically, a vertical configuration is
considered which does not allow for bypass flow left and right of the HATT, see Figure 20. In advance one can state that this theory will overestimate the blockage effects, because in practice there is bypass flow left and right of the HATT. The free surface proximity model is introduced in Whelan et al. [20].

The free-surface height drop and the performance of the disc are predicted by combining the continuity, Bernoulli and momentum equations. These equations are the base of the free-surface proximity model theory. For the derivation of these equations it is necessary to define the velocities in terms of the velocity \( u_0 \):

\[
\begin{align*}
u_w &= \alpha u_0, \quad u_d = \beta u_0 \quad \text{and} \quad u_2 = \tau u_0 \quad \text{see Figure 76} \\
\end{align*}
\]

Equation A.18

The blockage factor is defined as \( B = \frac{D}{h_1} \). Note that because of the blockage the flow \( (u_2) \) around the disc will accelerate, see Figure 76.

Combining the continuity equation on the bypass flow \( (u_2) \) and (separately evaluated) the continuity equation on the inside stream tube gives:

\[
\beta = \frac{\alpha}{B(\tau - \alpha)} \left[ \tau \left( 1 - \frac{\delta z}{h_1} \right) - 1 \right]
\]

Equation A.19

In which \( \delta z \) represents the free surface height drop, which can be obtained through the use of Bernoulli’s equation along the free surface:

\[
\frac{\delta z}{h_1} = \frac{Fr^2}{2} \left( \tau^2 - 1 \right)
\]

Equation A.20

Figure 76 Control volume with actuator disc and boundaries. Modified figure from [20].
In which \( Fr = \frac{U_0}{\sqrt{gh_1}} \). By applying the Bernoulli equation on the centre line on the upstream and downstream part the force (per unit of width) on the disc is given (similar to actuator disc theory):

\[
F = D\delta p = \rho D \left[ g\delta z + \frac{1}{2} u_0^2(1 - \alpha^2) \right] \tag{Equation A.21}
\]

Alternatively the force can be calculated by applying the momentum balance on the control volume \( r \), see Figure 20:

\[
F = \int_0^{h_1} (P_a + \rho g z)dz - \int_0^{h_2} (P_a + \rho g z)dz - P_a(h_1 - h_2) - \rho(h_2 - s_w)\tau U^2
\]

\[
= \frac{1}{2} \rho g(2h_1\delta z - (\delta z)^2) + \rho U^2 h_1(1 - \tau) + \rho \beta U^2 D(\tau - \alpha) \tag{Equation A.22}
\]

Combining Equation A.21 with Equation A.22 in combination with Equation A.19 and Equation A.20 gives a quartic polynomial in the change in velocity of the bypass wake flow \( \tau \):

\[
Fr^2 \tau^4 + 4\alpha Fr^2 \tau^3 + (4B - 4 - 2Fr^2)\tau^2 + (8 - 8\alpha - 4Fr^2 \alpha)\tau
\]

\[
+ (8\alpha - 4 + Fr^2 - 4\alpha^2 B) = 0 \tag{Equation A.23}
\]

By finding solutions for \( \tau \) in terms of \( \alpha \) it is possible to calculate various other parameters. Note that the following relations can be derived from the above equations:

\[
a = 1 - \beta \tag{Equation A.24}
\]

\[
C_t = (\tau^2 - \alpha^2) \tag{Equation A.25}
\]

In which \( a \) is the axial induction factor from actuator disc theory. Using Equation A.23 and the relations of Equation A.24 and Equation A.25 other velocities values (of \( u_d \) and \( u_w \)) are found compared to the actuator disc theory, the influence of the boundaries are included. These other velocity values are combined with blade element momentum theory to derive a new value of \( C_t \) for the blocked case.

The free-surface proximity model is introduced in Whelan et al. [20]. For details of the theory see Whelan et al. [20].
B. Self-similarity

This Appendix elaborates on the notion of self-similarity and gives the derivation of the self-similar solution of an axi-symmetrical wake.

Scaling of invariance (self-similarity) allows one to find all the invariant solutions of a partial differential equation. The solutions are found by stretching the invariances, it requires to find the terms (by inspection) that do not change the equation and the boundary conditions. The self-similarity solutions can be interpreted (after some change of variables) as the steady solution, in the limits of the solution will tend towards this steady solution. One might say that the solution is attracted towards this steady solution. 'the self-similar solution describes [68] the 'intermediate asymptotic' behaviour of solutions of wider classes of problems in the range where the solutions no longer depends on the details of the initial and/or boundary conditions’ [69]. Thus, self-similarity assumes that properly scaled quantities in a wake become independent of the scaled stream wise coordinate after some distance in axial direction [44]. A necessary condition for the existence of a similarity solution is that all scaled terms in the momentum equation are independent of the dimensionless stream wise coordinate x see Starke [44].

Self-similar solution of the axi-symmetrical wake

The planar wake of Figure 78 is extended to an axis symmetrical three dimensional wake see Figure 77. The undisturbed velocity \( u_0 \), and velocity at the centre line of the wake \( u_1 \) are the same as defined in Figure 8. The only difference is the addition of another axis (z axis) and the corresponding velocity \( w \) in this direction.

---

33 Momentum equation with zero pressure, dimensionless y coordinate, dimensionless u velocity, dimensionless v velocity and dimensionless shear stress and transverse velocity eliminated by using the mass equation.
The following derivation of the self-similarity solution for an axis symmetrical wake follows the work of Tennekes and Lumley [45] for a planar wake, only modified for an axi-symmetrical wake.

The cross stream length scale $\delta$ is defined as the distance from the centre line where the velocity deficit $(u_0 - u)$ is half the velocity deficit on the centre line of the wake $(u_0 - u_1)$, see Figure 78. The scale of change in the x direction is defined as L, see Figure 78. In general it is expected in wakes that the normalized velocity deficit $\frac{(u_0 - u)}{u_1}$ is a function of normalized y $(y/\delta)$ and z $(z/\delta)$ coordinate, normalized width of the wake $(\delta/L)$, wake Re number $(\delta u_1/\nu)$ and normalized velocity at the centre line of the wake $(u_1/u_0)$, see Equation B.1

$$\frac{(u_0 - u)}{u_1} = f(y/\delta, z/\delta, \delta/L, \delta u_1/\nu, u_1/u_0) \quad \text{Equation B.1}$$

However for the case that $\delta/L \to 0$, $\delta u_1/\nu \to \infty$ and $u_1/u_0 \to 0$ the function f is no longer depended on $\delta/x$, $\delta u_1/\nu$ and $u_1/u_0$, ‘because no monotone function can remain finite if it does not become asymptotically independent of very large or very small parameters’ [45]. Thus Equation B.1 can be written as:

$$\frac{(u_0 - u)}{u_1} = f(y/\delta, z/\delta) \quad \text{Equation B.2}$$

According to Tennekes H. and Lumley J.L. [45] in wakes the turbulent intensity is of the same order as the minimum velocity $u_1$. The Reynolds stress can thus be written as:

---

34 The velocity is normalized by the velocity on the centre line of the wake $u_1$, further in the thesis the velocity deficit is normalized by the undisturbed velocity $u_0$. However for this derivation it is more handy to normalize the velocity deficit by $u_1$.

35 Note that the turbulence which has been deemed as important for the wake recovery is represented by the magnitude of $u_1$ in the self-similarity solution.
\[ -\bar{u}'v' = u_1^2 g(y/\delta, z/\delta) \]

Equation B.3

In which \( u \) and \( v \) are the velocities in the \( x \) and \( y \) direction respectively, \( g \) is a function and the prime indicates the deviation of the velocity from its ensemble mean and the overbar indicates ensemble average. Equation B.2 and Equation B.3 constitute the self-similarity hypothesis.\(^{36}\) Their feasibility is tested by substituting them in the equation of motion. The streamwise momentum equation for \( u \) reads:

\[
u \frac{du}{dx} + v \frac{du}{dy} + w \frac{du}{dz} + \frac{\partial}{\partial x} (u^2 - v^2) + \frac{\partial}{\partial y} (u'v') + \frac{\partial}{\partial z} (u'w') = \frac{\partial}{\partial x} (\bar{v}'w')
\]

Equation B.4

In which \( \nu \) is the kinematic viscosity. Note that in Equation B.4 the pressure (\( P \)) term plus the Reynolds normal stress is zero, e.g. \( \frac{\partial}{\partial x} (v'w') + 1/\rho \frac{\partial \bar{P}}{\partial x} + \frac{\bar{v}^2}{\bar{v}^2} = 0 \) (in which \( \rho \) is the density of water) this relation has been obtained with the use of the cross stream momentum equation.

**Cross stream momentum equation**

This textbox shows how the cross stream momentum equation can be simplified to:

\[
\frac{\partial}{\partial x} (\bar{v}'w') + 1/\rho \frac{\partial \bar{P}}{\partial x} + \frac{\bar{v}^2}{\bar{v}^2} = 0
\]

Equation B.5

Start by defining \( u_1 = \max|u_0 - u| \) and in wakes \( u_1 \ll u_0 \) so that \( u = u_0 + (u - u_0) \approx O(u_0 + u_1) = O(u_0) \), in which \( O \) stands for the order of magnitude. Define cross stream length \( \delta \) as the distance where \( u - u_0 \approx 0.5u_1 \), thus \( \frac{du}{dy} \approx O(u_1/\delta) \) and define the scale of change in the \( x \) direction \( L \), see Figure 78.

Thus, \( \frac{du}{dy} = O(u_1/L) \), define \( \mu \) is the velocity scale for turbulence \( u'v' = O(\mu^2) \), \( u'^2 = O(\mu^2) \) and \( \nu^2 = O(\mu^2) \).

The continuity equation:

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0
\]

Equation B.6

Because \( \partial u/\partial x \sim u_1/\delta \), \( \partial v/\partial y \sim u_1/L \) and \( \partial w/\partial z \sim w/\delta \) which together with \( \partial v/\partial y \sim v/\delta \), \( \partial w/\partial z \sim w/\delta \) gives \( v = O(u_1 \delta/L) \) and \( w = O(u_1 \delta/L) \).

The cross stream momentum equation is given by:

\(^{36}\) ‘the velocity defect and the Reynolds stress become invariant with respect to \( x \) if they are expressed in term of the local length and velocity scales.’ 45. Tennekes, H. and J.L. Lumley, *A first course in turbulence*. 1972: MIT.
\[
u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} + \frac{\partial}{\partial y} \left( v'^2 \right) + \frac{\partial}{\partial x} \left( u' v' \right) + \frac{\partial}{\partial z} \left( v' w' \right) = -\frac{1}{\rho} \frac{\partial P}{\partial y} + v \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right)
\]

In which by inspection the order of magnitudes are

\[
u \frac{\partial v}{\partial x} = O \left( \frac{u_0 u_1 \delta}{L} \right) = \left[ \frac{u_0 u_1}{\mu} \left( \frac{\delta}{L} \right) \right]^2 \frac{\mu^2}{\delta}
\]
\[
u \frac{\partial v}{\partial y} = O \left( \left( \frac{u_1 \delta}{L} \right)^2 \frac{1}{\delta} \right) = \left[ \left( \frac{u_1}{\mu} \right)^2 \left( \frac{\delta}{L} \right) \right] \frac{\mu^2}{\delta}
\]
\[
u \frac{\partial v}{\partial z} = O \left( \left( \frac{u_1 \delta}{L} \right)^2 \frac{1}{\delta} \right) = \left[ \left( \frac{u_1}{\mu} \right)^2 \left( \frac{\delta}{L} \right) \right] \frac{\mu^2}{\delta}
\]
\[
\frac{\partial}{\partial x} (u' v') = O \left( \frac{\mu^2}{L} \right) = \frac{\delta \mu^2}{L \delta}
\]
\[
\frac{\partial}{\partial z} (w' v') = O \left( \frac{\mu^2}{\delta} \right) = \frac{\mu^2}{\delta}
\]
\[
\frac{\partial}{\partial y} (v'^2) = O \left( \frac{\mu^2}{\delta} \right) = \frac{\mu^2}{\delta}
\]
\[
1 \frac{\partial P}{\rho \partial y} = ?
\]
\[
u \frac{\partial^2 v}{\partial x^2} = O \left( \frac{v u_1 \delta}{L} \right) = \left[ \frac{u_1}{\mu Re} \left( \delta \right)^3 \right] \frac{\mu^2}{\delta}
\]
\[
u \frac{\partial^2 v}{\partial y^2} = O \left( \frac{v u_1 \delta}{L} \right) = \left[ \frac{u_1}{\mu Re} \left( \frac{\delta}{L} \right) \right] \frac{\mu^2}{\delta}
\]
\[
u \frac{\partial^2 v}{\partial z^2} = O \left( \frac{v u_1 \delta}{L} \right) = \left[ \frac{u_1}{\mu Re} \left( \delta \right)^3 \right] \frac{\mu^2}{\delta}
\]

In which \( Re = \frac{u \delta}{v} \). By inspection of the orders of magnitude the first, second, third, fifth and sixth terms of Equation B.7 can be neglected with respect to the \( \frac{\partial}{\partial y} (v'^2) \) term. [45] In sufficient high \( Re \) viscous effects can be neglected in Equation B.7. Because this thesis is only concerned with the wake in the turbulent regime, the \( Re \) is sufficiently high. Thus Equation B.7 can be simplified to:

\[
\frac{\partial}{\partial y} (v'^2) + \frac{\partial}{\partial z} (v' w') = -\frac{1}{\rho} \frac{\partial P}{\partial y}
\]

Equation B.9

Integration gives:

\[
(v' w') + \frac{P}{\rho} + \overline{v'^2} = P_0/\rho
\]

Equation B.10

The derivative with respect to x direction gives:

\[
\frac{\partial}{\partial x} (v' w') + 1/\rho \frac{\partial P}{\partial x} + \overline{v'^2} = 0
\]

Equation B.11
By evaluating the orders of magnitude Equation B.4 (see textbox below) can be simplified to

$$u_0 \frac{\partial u}{\partial x} + \frac{\partial u'v'}{\partial y} + \frac{\partial u'w'}{\partial z} = 0$$  \hspace{1cm} \text{Equation B.12}

**stream wise momentum equation**

This textbox shows how the stream wise momentum equation can be simplified to:

$$u_0 \frac{\partial u}{\partial x} + \frac{\partial u'v'}{\partial y} + \frac{\partial u'w'}{\partial z} = 0$$  \hspace{1cm} \text{Equation B.13}

The stream wise momentum equation is given by Equation B.14:

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} + \frac{\partial}{\partial x} (u^2 - v^2) + \frac{\partial}{\partial y} (u'v') + \frac{\partial}{\partial z} (u'w') - \frac{\partial}{\partial x} (v'w') = \nu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right)$$  \hspace{1cm} \text{Equation B.14}

Once again the orders of magnitudes of the terms are evaluated:

\[
\begin{align*}
\frac{u}{L} \frac{\partial u}{\partial x} &= O \left( \frac{u_1}{L} \right) = \left[ u_0 \frac{u_1}{L} \right] \frac{\mu^2}{\delta} \\
\frac{v}{L} \frac{\partial u}{\partial y} &= O \left( \frac{u_1}{L} \right) = \left[ \frac{u_1}{\mu} \right] \frac{\mu^2}{\delta} \\
\frac{w}{L} \frac{\partial u}{\partial z} &= O \left( \frac{u_1}{L} \right) = \left[ \frac{u_1}{\mu} \right] \frac{\mu^2}{\delta} \\
\frac{\partial}{\partial x} (u^2 - v^2) &= O \left( \frac{\mu^2}{L} \right) = \frac{\delta}{L} \frac{\mu^2}{\delta} \\
\frac{\partial}{\partial y} (u'v') &= O \left( \frac{\mu^2}{L} \right) = \frac{\mu^2}{\delta} \\
\frac{\partial}{\partial z} (w'u') &= O \left( \frac{\mu^2}{L} \right) = \frac{\mu^2}{\delta} \\
\frac{\partial}{\partial z} (v'w') &= O \left( \frac{\mu^2}{L} \right) = \frac{\delta}{L} \frac{\mu^2}{\delta} \\
\frac{\nu}{L^2} \frac{\partial^2 u}{\partial x^2} &= O \left( \frac{\nu u_1}{L^2} \right) = \left[ \frac{u_1}{\mu \delta} \right] \frac{\mu^2}{\delta} \\
\frac{\nu}{L^2} \frac{\partial^2 u}{\partial y^2} &= O \left( \frac{\nu u_1}{\delta^2} \right) = \left[ \frac{u_1}{\mu \delta} \right] \frac{\mu^2}{\delta} \\
\frac{\nu}{L^2} \frac{\partial^2 u}{\partial z^2} &= O \left( \frac{\nu u_1}{\delta^2} \right) = \left[ \frac{u_1}{\mu \delta} \right] \frac{\mu^2}{\delta}
\end{align*}
\]  \hspace{1cm} \text{Equation B.15}
Once again as with the transverse momentum equation this thesis is only concerned with turbulent wake so that the term on the right hand side of Equation B.14 can be neglected. In the limit as $\delta / L \rightarrow 0$ the fourth and seventh terms of Equation B.14 can be neglected. Thus at least one term should be the same order as $\frac{\partial}{\partial y} (u' \nu')$ and $\frac{\partial}{\partial z} (u' \nu')$ and the first term of Equation B.14 is larger than the second and third (because $u_0 > u_1$), thus:

$$\frac{u_0 u_1 \delta}{\mu \mu L} = O(1)$$

Equation B.16

It one sets $u_1 / \mu = O(1)$ and remember that one evaluates the limit of $\delta / L \rightarrow 0$ this implies $u_0 / \mu = O(L / \delta)$ which is the case in the far wake. [45] According to Tennekes H. and Lumley J.L. (1972) [45] in wakes the turbulent intensity is of the same order as $u_1$, thus $\mu = O(u_1)$. With $\mu = O(u_1)$ and $u_0 / \mu = O(L / \delta)$ one can neglect the second and third term (of Equation B.14) compared to the first term. Equation B.14 can thus be rewritten as:

$$\frac{\partial u}{\partial x} + \frac{\partial u' \nu'}{\partial y} + \frac{\partial u' \nu'}{\partial z} = 0$$

Equation B.17

And because $\frac{(u - u_0)}{u_0} = O \left( \frac{u_1}{u_0} \right) = O \left( \frac{\delta}{L} \right)$ the undifferentiated $u$ in Equation B.17 can be written as $u_0$, thus coming to the approximation of Equation B.18:

$$u_0 \frac{\partial u}{\partial x} + \frac{\partial u' \nu'}{\partial y} + \frac{\partial u' \nu'}{\partial z} = 0$$

Equation B.18

Using $\varepsilon = y / \delta$ and $\psi = z / \delta$ gives:

$$\frac{\partial u}{\partial x} = - \frac{\partial u_1}{\partial x} f + \frac{u_1 \partial \delta}{\delta} \frac{\partial f}{\partial x} + \frac{u_1 \partial \delta}{\delta} \frac{\partial \psi}{\partial x} \hat{f}$$

Equation B.19

$$\frac{\partial u' \nu'}{\partial y} = - \frac{u_1^2}{\delta} \hat{g}$$

Equation B.20

and

$$\frac{\partial u' \nu'}{\partial z} = - \frac{u_1^2}{\delta} \hat{g}$$

Equation B.21

In which dot represents differentiation to $\varepsilon$ and hat represents differentiation to $\psi$. The self-similarity of Equation B.2 and Equation B.3 is now substituted in Equation B.12 (with the definitions of Equation B.19, Equation B.20 and Equation B.21):

$$- \frac{u_0 \delta u_1}{u_1^2} \frac{\partial f}{\partial x} + \frac{u_0 \partial \delta}{u_1} \hat{f} + \frac{u_0 \partial \delta}{u_1} \frac{\partial \psi}{\partial x} \hat{f} = \hat{g} + \hat{g}$$

Equation B.22
It is required for \( f, \varepsilon, g \) and \( \theta \) to be constant. If the normalized profiles of the velocity deficit and the Reynolds stress are the same at all \( x \) (self-similarity). This thus leads to the following:

\[ \frac{\delta}{u_1^2} \frac{\partial u_1}{\partial x} = \text{constant and} \quad \frac{1}{u_1} \frac{\partial \delta}{\partial x} = \text{constant} \]  

Equation B.23

The general solution of Equation B.23 is \( \delta \sim x^n \) and \( u_1 \sim x^{n-1} \). An extra relation is needed to fix the solution of Equation B.23, this (extra) relation is obtained by the momentum integral see textbox below.

**momentum integral**

This textbox gives the momentum integral:

\[ \rho \int_{-\infty}^{\infty} u(u - u_0)dydz = M \]  

Equation B.24

Equation B.12 is a special case of Equation B.25:

\[ \frac{u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} + \frac{\partial u'v'}{\partial y} + \frac{\partial u'w'}{\partial z} = 0 \]  

Equation B.25

Thus every condition valid for Equation B.25 is valid for Equation B.12 as well. Subtracting \( u_0 \) from \( u \) (within the stream wise derivative) one obtains:

\[ \frac{u}{\partial x} + v \frac{\partial (u - u_0)}{\partial y} + w \frac{\partial (u - u_0)}{\partial z} + \frac{\partial u'v'}{\partial y} + \frac{\partial u'w'}{\partial z} = 0 \]  

Equation B.26

Subtracting \( u_0 \) is permitted because \( u_0 \) is not a function of a position. The continuity equation

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \]  

can be used to rewrite the first three terms of Equation B.26 resulting in:

\[ \frac{\partial [u(u - u_0)]}{\partial x} + v \frac{\partial [v(u - u_0)]}{\partial y} + w \frac{\partial [w(u - u_0)]}{\partial z} + \frac{\partial u'v'}{\partial y} + \frac{\partial u'w'}{\partial z} = 0 \]  

Equation B.27

For sufficiently large values of \( y \) and \( z \) \( u - u_0, u'v' \) and \( u'w' \) tend to 0. Thus integration of Equation B.27 with respect to \( y \) and \( z \) results in:

\[ \frac{\partial}{\partial x} \int_{-\infty}^{\infty} u(u - u_0)dydz = 0 \rightarrow \rho \int_{-\infty}^{\infty} u(u - u_0)dydz = M \]  

Equation B.28

With the definition that \( \chi \) is the area which represents the separated stagnant region of a (fictive) wake one can obtain a relation for the momentum flux \( M \). The net momentum per unit of volume is \( \rho u_0 \) (the wake does not contain momentum). The total volume per unit of time is \( u_0 \chi \), thus \( \rho u_0^2 \chi \) represents the
net momentum defect per unit time:

\[ M = -\rho u_1^2 \chi \]  \hspace{1cm} \text{Equation B.29}

The momentum integral is given as:

\[ \rho \int_{-\infty}^{\infty} u(u - u_0)dydz = M \]  \hspace{1cm} \text{Equation B.30}

Equation B.30 can be rewritten with the use of Equation B.2:

\[ u_0u_1 \delta^2 \int_{-\infty}^{\infty} f(\varepsilon, \psi)d\varepsilon d\psi - u_1^2 \delta^2 \int_{-\infty}^{\infty} f^2(\varepsilon, \psi)d\varepsilon d\psi = -\frac{M}{\rho} \]  \hspace{1cm} \text{Equation B.31}

The second term in Equation B.31 is of order \( u_1/u_0 \) compared to the first term \([45]\), with \( u_1/u_0 \) is of order \( \delta/L \), see textbox cross stream momentum equation. Note that we consider the case that \( \delta/L \rightarrow 0 \) so the second term is neglected:

\[ u_0u_1 \delta^2 \int_{-\infty}^{\infty} f(\varepsilon, \psi)d\varepsilon d\psi = -\frac{M}{\rho} \]  \hspace{1cm} \text{Equation B.32}

With the definition that \( \chi \) is the area which represents the separated stagnant region of a (fictive) wake one can obtain a relation for the momentum flux \( M \). The net momentum per unit of volume is \( \rho u_0 \)(the wake does not contain momentum). The total volume per unit of time is \( u_0 \chi \), thus \( \rho u_0^2 \chi \) represents the net momentum defect per unit time:

\[ M = -\rho u_1^2 \chi \]  \hspace{1cm} \text{Equation B.33}

With the help of Equation B.33, Equation B.32 can be written as:

\[ u_1 \delta^2 \int_{-\infty}^{\infty} f(\varepsilon, \psi)d\varepsilon d\psi = u_0 \chi \]  \hspace{1cm} \text{Equation B.34}

The product of \( u_1 \delta^2 \) should be independent of \( x \) and with the relations is \( \delta \sim x^n \) and \( u_1 \sim x^{n-1} \) this implies that \( 3n - 1 = 0 \rightarrow n = \frac{1}{3} \). Thus the self-similar solution states:

\[ \delta \sim x^{1/3} \]
\[ u_1 \sim x^{-2/3} \]  \hspace{1cm} \text{Equation B.35}

And velocity deficit \( u_{df}(y) = u_1(x)\exp[-y^2/(\lambda \delta^2)] \)
C. Steady state

This appendix elaborates on the steady state assumption for the assessment of the wake of the HATT. The goal of this Appendix is to check if the change of the ambient velocity is significant compared to the advection speed of the wake.

The Experiments of Stallard et al. [39] show measurements up to 20D downstream of the turbine. Therefore, it is assessed if the change of ambient velocity is significant in the time it takes for the wake to reach 20D downstream of the rotor.

For the assessment it is required to obtain the maximum acceleration of the ambient tidal current. Because, Marsdiep is a possible location for a tidal farm the maximum acceleration of the tidal current at Den Helder is taken to be normative. The maximum acceleration at Den Helder is $3.6015 \times 10^{-4} \text{ m/s}^2$ estimated from [56]. For the case of Stallard et al. [39] this means that the ambient tidal current velocity changes by 4.4% in the time it takes for a water particle travelling with the lowest measured velocity in the wake to reach 20D downstream of the turbine.

The assessment is based on the worst case scenario for the validity of the steady state assumption. The maximum acceleration of the tidal current velocity is used in combination with the smallest velocity measured in the wake. Thus, in the worst case scenario the velocity of the ambient tidal current changes by 4.4%, this can be considered small for a worst case scenario. It is reasonable to assume the steady state assumption for the assessment of the wake up to 20D downstream of the turbine.
D. Delft3D-FLOW

Delft3D-flow uses numerical approximation to solve the shallow water equations. The actual to solve equations are:

1. The mass balance equation
2. The horizontal momentum equations

The continuity equation is given in the σ-coordinates, which reads as follows:

\[
\frac{\partial \zeta}{\partial t} + \frac{\partial hu}{\partial u} + \frac{\partial hv}{\partial y} + \frac{\partial w}{\partial \sigma} = 0
\]

In which \( \zeta \) is the surface elevation and \( h \) is the total water depth. The continuity equation reads as follows:

\[
\sigma = \frac{z - \zeta}{h}
\]

Equation D.1

Equation D.2

Note that Equation D.2 does actually has to be zero as there may be a discharge or withdrawal of water due to precipitation and evaporation.

The horizontal momentum equation in the x direction reads as follows:

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + \frac{w \partial u}{h \partial \sigma} = - \frac{1}{\rho_0} P_x + f v + F_x + \frac{1}{h^2 \partial \sigma} \left( v y \frac{\partial u}{\partial \sigma} \right) + M_x
\]

Equation D.3

The horizontal momentum equation in the y direction reads as follows:

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + \frac{w \partial v}{h \partial \sigma} = - \frac{1}{\rho_0} P_y + f u + F_y + \frac{1}{h^2 \partial \sigma} \left( v x \frac{\partial v}{\partial \sigma} \right) + M_y
\]

Equation D.4

\( f = \) coriolis force per unit of mass [N/kg]

\( F_x \) and \( F_y \) represent the unbalance of horizontal Reynolds stresses.

\( v_t^x = \) horizontal eddy viscosity [m\(^2\)/s]

\( v_t^y = \) vertical eddy viscosity [m\(^2\)/s]

\( P_x = \) pressure term in x direction = \( \rho_0 \frac{\partial \zeta}{\partial x} + gh \int_{\sigma}^{0} (\frac{\partial p}{\partial x} + \frac{\partial \rho \partial \sigma}{\partial \partial x}) \, d\sigma' \)

\( P_y = \) pressure term in y direction = \( \rho_0 \frac{\partial \zeta}{\partial y} + gh \int_{\sigma}^{0} (\frac{\partial p}{\partial y} + \frac{\partial \rho \partial \sigma}{\partial \partial y}) \, d\sigma' \)

\( M_x = \) contribution due to external sources or sinks
M_y = contribution due to external sources or sinks

The vertical velocity is obtained from the continuity equation.

In Equation D.3 and Equation D.4 the Reynolds stresses are modelled (F_x and F_y) using the eddy viscosity concept. The horizontal Reynolds stresses are defined as:

\[ F_x = \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} \]  
\[ F_y = \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} \]

For small scale flow when the shear stresses at the closed boundaries are taken into account, the shear stresses are as follows:

\[ \tau_{xx} = 2v_H \left( \frac{\partial u}{\partial x} + \frac{\partial u}{\partial \sigma} \frac{\partial \sigma}{\partial x} \right) \]  
\[ \tau_{xy} = \tau_{yx} = v_H \left[ \left( \frac{\partial u}{\partial y} + \frac{\partial u}{\partial \sigma} \frac{\partial \sigma}{\partial y} \right) + \left( \frac{\partial v}{\partial x} + \frac{\partial v}{\partial \sigma} \frac{\partial \sigma}{\partial x} \right) \right] \]  
\[ \tau_{yy} = 2v_H \left( \frac{\partial v}{\partial y} + \frac{\partial v}{\partial \sigma} \frac{\partial \sigma}{\partial y} \right) \]

However, for large-scale flow with a coarse grid, when the shear stresses along the closed boundaries may be neglected, the forces F_x and F_y are simplified:

\[ F_x = \nu_t^H \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial x^2} \right) \]  
\[ F_y = \nu_t^H \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial x^2} \right) \]

Depending on the needs of the user, one can choose between the two sets of equations for F_x and F_y.

The horizontal eddy viscosity coefficient may be a superposition of three parts:

- a part due to “sub-grid scale turbulence”,
- a part due to “3D-turbulence”
- a part for depth-averaged simulations (includes molecular viscosity).[49]

A brief explanation of the three parts of the horizontal eddy viscosity
The horizontal eddy viscosity is mostly associated with the contribution of horizontal turbulent motions and forcing that are not resolved by the horizontal grid \(v_{SGS}\) or by the Reynolds-averaged shallow-water equations \(v_{H}^{back}\). The total horizontal eddy viscosity can thus be written as:

\[
v_H = v_{SGS} + v_V + v_{H}^{back}
\]

Equation D.12

If the option Horizontal large eddy simulation (HLES) is selected the sub-grid viscosity is determined as follows:

\[
v_{SGS} = \frac{1}{k_S^2} \left( \sqrt{(y\sigma_T S)^2 + B^2} - B \right)
\]

Equation D.13

In which \(\frac{1}{k_S^2} = \frac{\Delta x \Delta y}{(\pi t_{dp})^2} (f_{dp} \text{ is a constant depending on the numerical scheme})\), \(\gamma\) is the relation of the slope in the log-log spectrum of energy density as a function of the wave number magnitude, \(\sigma_T\) is the Prandtl-Schmidt number, \(B = \frac{3 g |U|}{4 H C^2}\) (\(C\) is the Chezy coefficient and \(H\) the water depth) and \(S\) the sum of the strain rates. For a detailed description of HLES see Delft3D-FLOW user manual [49].

The contribution of the 3D-turbulence is taken into account by \(v_V\). \(v_V\) is computed in the simulation using a 3D-turbulence model. The vertical eddy viscosity \(v_V\) is defined by:

\[
v_V = v_{mol} + max(v_{3D}, v_{V}^{back})
\]

Equation D.14

\(v_{mol}\) is the kinematic viscosity of water. \(v_{3D}\) is the part computed by the 3D-turbulence model and \(v_{V}^{back}\) is the ambient vertical mixing coefficient set by the user.

**Turbulence model**

Delft3D-flow has four turbulence closure models to determine \(v_V\) and the vertical diffusion coefficient \(D_V\):

1. constant coefficient
2. Algebraic Eddy viscosity closure model (AEM)
3. \(k-L\) turbulence closure model
4. \(k-\epsilon\) turbulence closure model

Only the \(k-\epsilon\) turbulence closure model will be discussed. The \(k-\epsilon\) turbulence closure model is based on the eddy viscosity concept, the eddy viscosity is related to a characteristic length and velocity scale:

\[
v_{3D} = c'_\mu L \sqrt{k}
\]

Equation D.15

In which \(c'_\mu = 0.09\) a constant determined by calibration (empirical), see Delft3D-FLOW manual [49]

L = mixing length and

\(k\) = turbulent kinetic energy.
The k-ε turbulence closure model requires \( L \) to be determined from \( k \) and \( \varepsilon \) (dissipation). \( L \) is related to these as follows:

\[
L = c_D \frac{k \sqrt{k}}{\varepsilon}
\]  

Equation D.16

A transport equation for \( k \) and \( \varepsilon \) must be solved, however the following assumptions have been made:

- The production, buoyancy and dissipation terms are the dominating terms.
- The horizontal length scales are larger than the vertical ones.

The transport equations for \( k \) and \( \varepsilon \):

\[
\frac{\partial k}{\partial t} + u \frac{\partial k}{\partial x} + v \frac{\partial k}{\partial y} + \frac{w}{h} \frac{\partial k}{\partial \sigma} = \frac{1}{h^2} \frac{\partial}{\partial \sigma} \left( D_k \frac{\partial k}{\partial \sigma} \right) + p_k + p_{kw} + B_k - \varepsilon
\]

Equation D.17

\[
\frac{\partial \varepsilon}{\partial t} + u \frac{\partial \varepsilon}{\partial x} + v \frac{\partial \varepsilon}{\partial y} + \frac{w}{h} \frac{\partial \varepsilon}{\partial \sigma} = \frac{1}{h^2} \frac{\partial}{\partial \sigma} \left( D_\varepsilon \frac{\partial \varepsilon}{\partial \sigma} \right) + p_\varepsilon + p_{\varepsilon w} + B_\varepsilon - c_2 \frac{\varepsilon^2}{k}
\]

Equation D.18

In which \( D_k = \frac{v_{mol}}{\sigma_{mol}} + \frac{v_{3D}}{\sigma_k} \) and \( D_\varepsilon = \frac{v_{3D}}{\sigma_\varepsilon} \)

Equation D.19

In the production term \( p_k \), the vertical velocity gradient is neglected with respect to gradients of horizontal velocities as is the curvature of the \( \sigma \)-grid:

\[
p_k = 2v_{3D} \left[ \frac{1}{2h^2} \left( \frac{\partial u}{\partial \sigma} \right)^2 + \left( \frac{\partial v}{\partial \sigma} \right)^2 \right] + 2v_{3D} \left[ \frac{1}{2} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial v}{\partial x} \right)^2 \right]
\]

Equation D.20

\[
p_\varepsilon = c_1 \frac{\varepsilon}{k} p_k
\]

Equation D.21

The buoyancy term \( B_k \) is a result of the conversion of turbulent kinetic energy into potential energy in stratified flows:

\[
B_k = \frac{v_{3D} \rho g \partial \rho}{\rho \sigma_p h \frac{\partial \sigma}{\partial \sigma}}
\]

Equation D.22

\[
B_\varepsilon = c_1 \frac{\varepsilon}{k} (1 - c_3 \varepsilon) B_k
\]

Equation D.23

In which \( \sigma_p \) is the Prandl-Smidt number.

The production and dissipation terms \( p_{kw} \) and \( p_{\varepsilon w} \) in Equation D.17 and Equation D.18 are terms who account for waves. The vertical eddy viscosity \( v_{3D} \) is determined by:

\[
v_{3D} = c_\mu L \sqrt{k} = c_\mu k^2
\]

Equation D.24

In which \( c_\mu = c_D c_\mu \)

Dirchlet boundary conditions are used to solve the transport equations.
**Vertical accelerations: non-hydrostatic mode**

The hydrostatic assumption is valid if the vertical acceleration can be neglected compared to the gravitational acceleration. This section gives an example calculation of the vertical acceleration at the inflow based on the data of Stallard et al. [39].

The vertical acceleration is defined as:

\[
\frac{Dw}{Dt} = \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z}
\]  

Equation D.25

Inspection of Equation D.25 is performed on the flow just in front of the turbine, see Figure 79.

Therefore, the terms of Equation D.25 are evaluated:

- The steady state is considered. Consequently, the first term on the right hand side \(\frac{\partial w}{\partial t}\) is zero.
- The second term \(u \frac{\partial w}{\partial x}\). The value of \(u\) just in front of the turbine \((u = u_d)\) can be calculated based on the actuator theory of section 2.3.1. The vertical velocity difference \(\partial w\) can be approximated by using the conservation of energy and assuming that the initial vertical velocity is zero. Note that from symmetry it follows (if the considered vertical lengths are small) that the vertical velocity and horizontal velocity in the y direction are equal in value, e.g. \(v = w\), see Figure 79.
  
  Conservation of energy applied to stream line 2 (see Figure 79) gives:

\[
\frac{1}{2} \rho u_0^2 + \rho gh = \frac{1}{2} \rho u_d^2 + \rho gh + \frac{1}{2} \rho v^2 \rightarrow v = \sqrt{u_0^2 - u_d^2}
\]
  
  Equation D.26

Thus, the velocity difference \(\Delta w = w - 0 = \sqrt{u_0^2 - u_d^2}\). To estimate the gradient it is required to estimate the length in the x direction in which this velocity difference occurs. Roc et al. [36] have performed numerical simulations\(^{37}\) of HATTs showing that the velocity change in front of the turbine for the most part occurs within 0.5D. Thus, the gradient \(\frac{\partial w}{\partial x}\) and consequently the value of the second term can be estimated.
- The third \((v \frac{\partial w}{\partial y})\) and fourth \((w \frac{\partial w}{\partial z})\) term. The vertical and horizontal velocity and gradients can be considered equal due to symmetry of the flow (see Figure 79), e.g. \(v = w\) and \(\frac{\partial w}{\partial y} = \frac{\partial w}{\partial z}\), thus \(v \frac{\partial w}{\partial y} = w \frac{\partial w}{\partial z}\). The inspection of the second term has provided an expression for the vertical w and horizontal v velocity. Consequently, the continuity equation applied just in front of the turbine can be simplified:

\(^{37}\) Information from literature about the characteristics of the flow in front of the turbine is scarce. It is preferred not to use numerical simulation for the determination of the velocity gradient just in front of the disc as the grid size can greatly influence the steepness of the gradients. However since measurements are not available the data of Roc et al. [36] is used.
The velocity gradient $\frac{\partial u}{\partial x}$ can be approximated using actuator disc theory (see explanation of the second term). Thus, the value of the velocity gradient $\frac{\partial w}{\partial z}$ can be estimated and consequently the third $(v \frac{\partial w}{\partial y})$ and fourth $(w \frac{\partial w}{\partial z})$ term of Equation D.26 can be estimated.

![Figure 79 schematic view of streamlines at the inflow of the turbine, turbine is displayed as an circle.](image)

The method of estimating the vertical accelerations is based on a number of assumptions. The validity of the method can be questioned, however in this case one is interested in the order of magnitude, not the exact value of the vertical acceleration.

The method described above is applied to the data of Stallard et al. [39]. The vertical acceleration of the inflow in the case of Stallard et al. [39] is:

$$\frac{Dw}{Dt} \approx 1.20 \text{ m/s}^2$$

Equation D.28

The vertical accelerations at the inflow are approximately 12% of the gravitational acceleration and thus cannot be neglected. From literature such as Carmer [41] it is known that in the near wake the hydrostatic assumption is not valid. However in the far wake the hydrostatic assumption is valid [41].

Delft3D-FLOW has a feature which allows the user to run models in non-hydrostatic mode. This mode uses a modification of the hydrostatic assumption, this modification is combined with the horizontal momentum equations. The standard hydrostatic assumption in delft3D is:
\[
\frac{\partial p}{\partial \sigma} = -g \rho h \rightarrow \text{integrated over depth:} \quad P = P_{\text{atm}} + gh \int_{\sigma}^{0} \rho(x, y, \sigma') d\sigma'
\]

In non-hydrostatic mode (only available in the Z-grid model coordinates) the pressure \( P \) integrated over depth is given by:

\[
P = P_{\text{atm}} + g \int_{\xi}^{\zeta} \rho dz' + q
\]

In which \( q \) is the non-hydrostatic part. The momentum equation in the \( z \) direction is given by:

\[
\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = - \frac{1}{\rho} P_{w} + F_{w} + \frac{1}{h^2} \frac{\partial}{\partial z} \left( \nu \frac{\partial w}{\partial z} \right)
\]

In which \( F_{w} = v_{t}^{H} \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial z^2} \right) \). Non-hydrostatic mode is used when the vertical accelerations cannot be neglected compared to the gravitational acceleration. It is preferred to use non-hydrostatic mode when modelling HATTS.

**Grid layer**

There are two ways to define the vertical grid: sigma-layer model and Z-layer model. Delft3d-flow uses sigma-layer model as the default way to define a grid. See Figure 80 for the definition of the sigma-layer model.

![Figure 80 Sigma plane grid](image)

The accuracy of the discretization of the vertical exchange processes is determined by the vertical grid system. The sigma-layer model gives significant errors in the approximation of strictly horizontal density.
gradients in areas with steep bottom topography [49]. A vertical grid (Z-grid) was introduced to cope with the short comings of the sigma-layer model. The Cartesian Z-grid model has horizontal co-ordinate lines see Figure 81.

![Figure 81 irregular representation of bottom boundary layer in the Z-model.](image)

Delft3D-flow uses the alternating direction implicit (ADI) schemes to solve the equations. For a more detailed description of Delft3D-FLOW see the Delft3D-FLOW user manual [49].
E. HATT implementation

This Appendix elaborates on the implementation of the HATT in Delft3D. Firstly, additional information about the definition of the HATT on the grid is presented. Secondly, attention is given to the determination of the quadratic friction coefficient. Finally, the calibration procedure of the selected method for the determination of the quadratic friction coefficient is present.

**HATT definition in Delft3D**

This section elaborates on the definition of the HATT in Delft3D. The following description is from the unpublished work of Deltares [70].

The HATT is schematized as a flat porous disc with negligible thickness. The disc can be defined as combinations of sub and super grid in the vertical and horizontal directions, see Figure 82. It is assumed that the disc has a circular shape. If the disc consists out of multiple grid cells the interfaces are assumed to be rectangles, the slope of sigma-layer planes are ignored. The intersection of the rectangle and circular disc is a part of a circle. The surface of the interface is computed as the sum of one quarter circle. The intersection can be empty or given by the sum (or subtraction) of usually one circle segment, two triangles and two rectangles.

![Figure 82 schematized view of the HATT on the grid](image)

The orientation of the disc is given by the angle $\alpha$ between the positive x axis and the direction of the positive flow direction, see Figure 83.

![Figure 83 Definition sketch of angle $\alpha$](image)
The reference velocity is taken a user set distance in front or behind the turbine. If the velocity in front and behind the turbine show the same sign (direction) than the upstream velocity is taken, if the sign differs the average is taken. Only the component normal to the turbine is taken into account.

**Accuracy quadratic friction coefficient**

This section elaborates on the different methods to calculate the quadratic friction coefficient. Three options to calculate the quadratic friction coefficient are investigated:

- With the use of the actuator disc theory (see Paragraph 2.3)
- With the use of the free-surface proximity model (see Paragraph 2.3)
- Calibration method

The quadratic friction coefficient is calculated with Equation 4.4:

\[
 c_{loss} = \frac{1}{2} \cdot C_t(u_0) \cdot \frac{u_0^2}{u_d} \quad \text{Equation 4.4}
\]

The three theories are used to calculate the quadratic friction coefficient for the case of the experiments of Stallard et al. [39] and used as input value for Delft3D. Thereafter, the force according to the Delft3D model is calculated:

- Based on the actuator disc theory the velocity at the disc \( u_d = 0.32 \text{ m/s} \). Thus, the quadratic friction coefficient \( c_{loss} \) based on Equation 4.4 is 0.94. This value of the quadratic friction coefficient has been used as an input value in the Delft3D model. Subsequently, the force on the disc is calculated by the summation of the contribution of every individual grid cells according to Equation E.1.

\[
 F = \rho \cdot A \cdot c_{loss} \cdot |\vec{u}_d| \vec{u}_d \quad \text{Equation E.1}
\]

In which the tilde stands for velocities obtained from the Delft3D model.

- Based on the free-surface proximity model the velocity at the disc \( u_d = 0.43 \text{ m/s} \). Thus, the quadratic friction coefficient \( c_{loss} \) based on Equation 4.4 is 0.52. This value of the quadratic friction coefficient has been used as an input value in the Delft3D model. Subsequently, the force on the disc is calculated by the summation of the contribution of every individual grid cells according to Equation E.1.

- The force according to the calibration method is calculated based on Equation 4.5. The reference velocity is taken 4 rotor diameters upstream of the disc.

The results are presented in Table 9.

<table>
<thead>
<tr>
<th>Force according to Stallard et al. [39]</th>
<th>Force according to actuator disc theory</th>
<th>Force using free-surface proximity model.</th>
<th>Force according to calibration method</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.50 ± 0.32 N</td>
<td>6.02 N</td>
<td>4.23 N</td>
<td>5.54 N</td>
</tr>
</tbody>
</table>

Table 9 Thrust force.
The calibration method models the force the most accurate. Therefore, additional check on the method is performed before the method is implemented in Delft3D.

Firstly, errors due to numerical scheme are investigated. The calibration method prescribes the force on the disc according a reference velocity, see section 4.3.2. Any difference in the prescribed force and actual simulated force are due to numerical scheme. Figure 84 shows the imposed and simulated thrust force. The errors due to the numerical scheme are negligible.

![Image](image.png)

**Figure 84** The thrust force on the disc.

Secondly, the force on the disc is calculated using the hydraulic head difference of the flume with and without a rotor.

<table>
<thead>
<tr>
<th>Output of Delft3D, imposed thrust force.</th>
<th>Force on the disc calculated by hydraulic head difference</th>
<th>Difference in %</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.54 N</td>
<td>5.76 N</td>
<td>≈ 4%</td>
</tr>
</tbody>
</table>

Table 10 Thrust force.

The difference between the output of Delft3D and the force calculated by using the hydraulic head should be the same. The difference is not small enough to be neglected 4%. The difference is likely caused by the discretization error and error in the calculation of the hydraulic head. Figure 85 shows that a coarser mesh will give a larger error. The hydraulic head is based on the depth averaged velocity. The shape factor $\beta$, used to calculate the hydraulic head from the depth averaged velocity is assumed to be one, see Equation E.

---

38 The deviation of the force equality method with the data of Stallard et al. [39] is due to differences in the reference velocity used in the force equality method.
\[ H = h + \frac{1}{d} \int_{z_b}^{z_0} \frac{u^2}{2g} dz = h + \beta \frac{U^2}{2g} \]  

Equation E.2

In which \( H \) is the hydraulic head, \( d \) the water depth \( z_b \) the \( z \) coordinate of the bottom, \( z_0 \) the \( z \) coordinate of the free-surface, \( u \) the velocity in stream wise direction, \( g \) gravitational acceleration and \( U \) the depth averaged velocity in the stream wise direction. Tuning of the shape factor to the shape found in the simulated flume would increase the accuracy of the force calculation.

![Figure 85](image)

Figure 85 relative difference between force calculated by using the hydraulic head and imposed thrust force by Delft3D.

The calibration method is the most accurate method. This method is implemented in Delft3D.

**Additional Calibration procedure**

This section describes the additional calibration procedure which may be necessary if there is no appropriate position from which the reference velocity can be extracted.

The goal of the additional calibration procedure is to investigate what the velocity at the HATT \( u_d \) is with the used model settings. A simple model definition consisting of the same model settings is used to obtain the velocity at the HATT \( u_d \). Once the velocity at the HATT is known with the used model settings, the quadratic friction coefficient \( C_{td} \) is calculated\(^{39}\) so that the left hand side of Equation 4.6 equals the right hand side. The calibration procedure should be repeated for various flow velocities. Consequently, one obtains a table which contains the undisturbed velocity and the corresponding quadratic friction coefficient \( C_{td} \). This table can be used as input for the actual model definition. Consequently, in the

\(^{39}\) This is calculated by Delft3D if the reference velocity equals the undisturbed velocity.
actual model definition the reference velocity is set at the HATT, e.g. $u_{\text{ref}} = u_d$. This method allows the thrust coefficient to be incorporated into the input of Delft3D-FLOW.

The calibration of the power coefficient requires a similar (simpler) method of calibration as the thrust force. A simple model definition consisting of the same model settings is used to obtain the velocity at the HATT $u_d$. Once the velocity at the HATT is known with the used model settings, the power coefficient is calculated so that the Equation 2.3 equals Equation 4.7. The calibration procedure should be repeated for various flow velocities. Consequently, one obtains a table which contains the undisturbed velocity and the corresponding power coefficient. This table can be used as input for the actual model definition.

*Calibration of the case study*

The HATT of the case study has been subjected to the additional calibration procedure as it is not clear from where the reference velocity could be taken in the case study.

The HATT is situated in the middle of a 2 km wide and 5 km long flume with about the same depth as the case study and with mesh size approximately the same as the case study (20 x 20 x 2.5 m). The thrust and power coefficient of the HATT are assumed constant. Hence, the HATT has been calibrated for only one undisturbed inflow velocity 0.6 m/s.

The reference velocity is taken 9 rotor diameters upstream of the turbine and the thrust coefficient is set to 0.8. Subsequently, the matching quadratic friction coefficient 0.48 is read from the Delft3D output. To calibrate the power coefficient, the velocity $u_d = 0.59$ at the disc is read from the Delft3D output. Subsequently, the matching power coefficient for the velocity at the disc is calculated $C_p = 0.39$.

These values of the thrust coefficient 0.96 (2 times the quadratic friction coefficient see Equation 4.4) and power coefficient 0.39 have been used as input files for the case study. Consequently, the reference velocity of the HATTs of the case study have been taken at the disc, e.g. $u_{\text{ref}} = u_d$. 

27
F. Turbulence kinetic energy
The data of Stallard et al. [39] lacks information about the cross stream turbulence intensities. This Appendix elaborates on the assumptions made to determine the cross stream turbulence intensities.

Nezu and Nakagawa [53] have compared multiple measurements of turbulent intensities of fully developed open channel flows. From these measurements they concluded that the stream wise turbulence intensity in kinetic energy is dominant and is approximately 55% of the total turbulent kinetic energy. In Uijttewaal and Booij [54] the same order of magnitude of the contribution of stream wise turbulent intensity is found. In Koziol [55] reference is made to a paper of Grinval and Nikora [71] which also show the same order of magnitude of the contribution of stream wise turbulent intensity for rivers. At the cross sections x=6 m and x=7 m the flow in the flume of Stallard et al. [39] is not fully developed. Thus, the contribution of stream wise turbulent intensity is taken to be 60% of the total turbulent kinetic energy. The assumption allows the turbulent kinetic energy \( k \) to be rewritten into turbulent intensities presented in Stallard et al. [39]. Consequently, the turbulent intensities of Stallard et al. [39] can be compared to values in Delft3D.

In the experiments of Stallard et al. [39] the stream wise turbulent intensity \( Tl_{x} \) is given:

\[
Tl_{x} = \frac{\sqrt{(u - \bar{u})^2}}{u_0} = \frac{\sqrt{\left(\bar{u}_x + u'_x - \bar{u}_x\right)^2}}{u_0} = \frac{\bar{u}_x^2}{u_0}
\]  

Equation F.1

The stream wise turbulent intensity \( Tl_{x} \) is related to the turbulence kinetic energy \( k \):

\[
k = \frac{1}{2} \left( Tl_{x}^2 \cdot u_0^2 + u'_x^2 + u'_z^2 \right)
\]  

Equation F.2

And the turbulence intensity \( I \) is related to the turbulence kinetic energy \( k \) by Equation F.3.

\[
k = \frac{1}{2} I^2 V^2
\]  

Equation F.3

In which \( V \) is the magnitude of the velocity. Consequently, with the assumption that the contribution of stream wise turbulent intensity is 60% of the total turbulent kinetic energy the stream wise turbulence intensity can be given by Equation F.4.

\[
Tl_{x} = \sqrt{\frac{1.2k}{u_0^2}}
\]  

Equation F.4
G. Model definition details

*Delft3D_1rotor*

Delft3D_1rotor has the same configuration as the experiment of Stallard et al. [39], see Figure 86. The mesh exists of 202 grid cells (including two dummy cells) in the y-direction, leading to a mesh size of \((\Delta y)\) 0.025 m. In the x-direction there are 482 grid cells (including two dummy cells), thus giving a mesh size of \((\Delta x)\) 0.025 m. The grid is made out of 31 layers in the z-direction, every layer has a mesh size of 0.015 m (3.125% of depth) except the two layers at the bottom and the top layer. The two bottom layers have a mesh size of 0.0075 m (1.5625% of depth). The top layer has a mesh size of 0.045 m (9.375% of depth). The grid cells in z-direction are set to cope with the large gradients at the bottom. The Z-model is used with the bottom at -0.45 m and the maximum level 0.03 m. The time step \((\Delta t)\) is set to 0.015 s, this small time step is chosen to meet the requirements set for the courant number, see Delft3D-Flow manual [49]. The total simulation time is 5 minutes, thus giving the total number of time steps to be 20000. Coriolis force is not taken into account, because Delft3D_1rotor tries to model the experiments of Stallard et al. [39], which are too small for coriolis force to have an significant impact.

The initial conditions are uniform and set to a water level of 0 m (thus meaning a water depth of 0.45 m). At the inflow (left side of Figure 86) a discharge per cell type of boundary is used. The type of forcing used is a time series. Time series is used to reduce spin up time of the model. A reflection parameter alpha of 5.7 s is used. The inflow increases linearly for the first 1.5 minutes from zero to 0.0053 m\(^3\)/s per horizontal grid cell, this leads to a total discharge of 1.06 m\(^3\)/s and an inflow velocity \(u_0\approx0.47\) m/s. The other boundary (right side see Figure 86) a water level boundary is set. The forcing type is time series
and a reflection parameter alpha of 1.2 s is used. The water level is set to 0 m leading to a total water depth of 0.45 m (as bathymetry is defined 0.45 m below reference level). The reflection parameters are used to lose the spin up phenomena faster.

The gravitational acceleration is set to 9.81 m/s$^2$ and the density of water to 1000 kg/m$^3$. It is not possible to impose a no slip condition to the bottom to simulate the roughness of the supposed glass bottom so instead a White-Colebrook bottom roughness coefficient is used of 5*10^-6 for both u and v velocities. Free slip conditions have been employed at the wall. A uniform background horizontal viscosity of 1*10^-6 m$^2$/s is used, this value has been obtained by scaling down the horizontal background viscosity of 1 m$^2$/s for a grid size of $\Delta x=\Delta y=25$m to the grid size used in this model definition ($\Delta x=\Delta y=0.025$m). The k-ε model is used for the 3D turbulence.

The depth is defined at the corners of the grid cells, at the middle of the grid cell the depth is taken to be the maximum of the value and at the faces of the grid cell the minimum value is used. The smoothing time is 0 min. Delft3D-FLOW allows the user to choose between multiple momentum solvers. In Delft3D_1rotor Mdui is used as momentum solver. Mdui stands for implicit multi-direction upwind discretisation. Mdui is an iterative method for solving the horizontal advection, an implicit method for the horizontal advection. The vertical advection is solved with central implicit method, see mymathlib.com for more information concerning central implicit method [72]. For the time integration the ADI scheme is used.

Additional parameters
Because it is expected that vertical accelerations cannot be neglected in the wake non-hydrostatic mode is used and thus Z-layer model (non-hydrostatic mode is only available in the Z-layer model). The maximum number of iteration of the momentum solver is set to 200. The absolute and relative convergence criterion is set to 1*10^-8. The pressure correction in non-hydrostatic mode is solved using the conjugate gradient method (CG) solver, see Delft3D-FLOW manual [49] for details. The implicit factor (θ) of the non-hydrostatic and the hydrostatic mode is set to one, this implicit factor refers to the time integration of the continuity equation. Near the porous plate upwind advection scheme is used. To simulate the turbulent conditions of Stallard et al. [39] at the inflow boundary, boundary conditions for the k-ε model are introduced. The turbulent boundary conditions at the inflow boundary are $k=0.08$ m$^2$/s$^2$ and $\varepsilon=0.0576$ m$^2$/s$^2$, so that $\nu_3d=10^{-2}$ m$^2$/s. Due to the turbulent boundary conditions the turbulent intensity in Delft3D_1rotor are similar to the turbulent intensities found in the experiments of Stallard et al. [39].

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Value/setting</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Zmodel</td>
<td>#Y#</td>
<td>z-model is used</td>
</tr>
<tr>
<td>Zbot</td>
<td>-0.45</td>
<td>Maximum depth of model</td>
</tr>
<tr>
<td>Ztop</td>
<td>0.03</td>
<td>Maximum water level</td>
</tr>
<tr>
<td>Nonhyd</td>
<td>#full#</td>
<td>Non-hydrostatic mode</td>
</tr>
<tr>
<td>Nhiter</td>
<td>200</td>
<td>maximum number of iteration of the momentum solver.</td>
</tr>
<tr>
<td>Epsnh</td>
<td>1e-8</td>
<td>Absolute convergence criterion</td>
</tr>
<tr>
<td>Repsnh</td>
<td>1e-8</td>
<td>Relative convergence criterion</td>
</tr>
<tr>
<td>Krylov</td>
<td>#cg#</td>
<td>solver</td>
</tr>
</tbody>
</table>
The use of the position of free surface for the determination of pressure gradient.

<table>
<thead>
<tr>
<th>Flagpp</th>
<th>#N#</th>
<th>The use of the position of free surface for the determination of pressure gradient.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Milu</td>
<td>0.5</td>
<td>Parameter of preconditioner</td>
</tr>
<tr>
<td>Tetaq</td>
<td>1.0</td>
<td>Implicity factor of non-hydrostatic pressure</td>
</tr>
<tr>
<td>Tetaz</td>
<td>1.0</td>
<td>Implicity factor of hydrostatic pressure</td>
</tr>
<tr>
<td>Filppl</td>
<td>#Turbine_1x1_v6.ppl#</td>
<td>File containing the porous plate characteristics</td>
</tr>
<tr>
<td>Upwppl</td>
<td>#Y#</td>
<td>Use of upwind scheme near porous plate</td>
</tr>
<tr>
<td>Nprcus</td>
<td>1</td>
<td>Number of user defined functions (one in this case)</td>
</tr>
<tr>
<td>Prcusr</td>
<td>#bc turbulence model#</td>
<td>Boundary conditions for k-ε model</td>
</tr>
<tr>
<td>Nprinp</td>
<td>1 0 0 0</td>
<td>One filename will be specified</td>
</tr>
<tr>
<td>Filusr</td>
<td>#turbound_kep-new.tbn#</td>
<td>File with boundary conditions for turbulence.</td>
</tr>
</tbody>
</table>

Table 11 overview of additional parameters

The porous disc has its centre at (6, 0, 0.225) respective x, y and z coordinates, similar to the experiments of Stallard et al. [39], see Figure 86. The diameter of the disc is 0.27 m and the thrust and power coefficient are 0.87 and 0.45 (independent on the flow velocities) respectively. The reference velocity from which the power and imposed thrust force is calculated are taken at the disc, with a corresponding quadratic friction coefficient of 0.75. The force of on the disc is approximately equal to the force on the rotor measured by Stallard et al. [39], see Appendix E.

**Delft3D_norotor**

The model definition Delft3D_norotor has the same characteristics as Delft3D_1rotor, with one exception. There is no rotor present in the flume.

**Delft3D_1rotor_support**

The model definition Delft3D_1rotor_support has the same characteristics as Delft3D_1rotor with the exception of existence of the support structure and the location of the reference velocity.

The support structure has been defined as a vertical plate with a width of 2.5 cm and reaches from the free surface to a depth of 0.255 m (z = -0.03 m). The plate is located 5 cm behind the rotor.

The reference velocity is taken 4 rotor diameters upstream of the disc and the thrust coefficient has been set to 0.87.

**Delft3D_2rotor**

The model definition Delft3D_2rotor has the same characteristics as Delft3D_1rotor, with one exception. There are two porous disc present at locations:

<table>
<thead>
<tr>
<th>Disc</th>
<th>x location [m]</th>
<th>y location [m]</th>
<th>z location [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6</td>
<td>0</td>
<td>-0.225</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>0.405</td>
<td>-0.225</td>
</tr>
</tbody>
</table>

Table 12 Location of porous discs.

The porous discs are 1.5D laterally spaced.
**Delft3D_3rotor**
The model definition Delft3D_3rotor has the same characteristics as Delft3D_1rotor, with one exception. There are three porous disc present at locations:

<table>
<thead>
<tr>
<th>Disc</th>
<th>x location [m]</th>
<th>y location [m]</th>
<th>z location [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6</td>
<td>-0.405</td>
<td>-0.225</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>0</td>
<td>-0.225</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>0.405</td>
<td>-0.225</td>
</tr>
</tbody>
</table>

Table 13 Location of porous discs.

The porous discs are 1.5D laterally spaced.

**Delft3D_3rotor2D**
The model definition Delft3D_3rotor2D has the same characteristics as Delft3D_3rotor, with one exception. The porous disc are spaced 2D in lateral direction. The locations of the discs are presented in Table 14.

<table>
<thead>
<tr>
<th>Disc</th>
<th>x location [m]</th>
<th>y location [m]</th>
<th>z location [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6</td>
<td>-0.54</td>
<td>-0.225</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>0</td>
<td>-0.225</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>0.54</td>
<td>-0.225</td>
</tr>
</tbody>
</table>

Table 14 Location of porous discs.

**Delft3D_3rotor3D**
The model definition Delft3D_3rotor3D has the same characteristics as Delft3D_3rotor, with one exception. The porous discs are spaced 3D in lateral direction. The locations of the discs are presented in Table 15.

<table>
<thead>
<tr>
<th>Disc</th>
<th>x location [m]</th>
<th>y location [m]</th>
<th>z location [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6</td>
<td>-0.81</td>
<td>-0.225</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>0</td>
<td>-0.225</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>0.81</td>
<td>-0.225</td>
</tr>
</tbody>
</table>

Table 15 Location of porous discs.
**Delft3D_5rotor**

The model definition Delft3D_5rotor has the same characteristics as Delft3D_1rotor, with one exception. There are five porous discs present at locations:

<table>
<thead>
<tr>
<th>Disc</th>
<th>x location [m]</th>
<th>y location [m]</th>
<th>z location [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6</td>
<td>-0.81</td>
<td>-0.225</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>-0.405</td>
<td>-0.225</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>0</td>
<td>-0.225</td>
</tr>
<tr>
<td>4</td>
<td>6</td>
<td>0.405</td>
<td>-0.225</td>
</tr>
<tr>
<td>5</td>
<td>6</td>
<td>0.81</td>
<td>-0.225</td>
</tr>
</tbody>
</table>

Table 16 Location of porous discs.

The porous discs are 1.5D laterally spaced.

**Model definitions of the sensitivity analysis**

In the sensitivity analysis of chapter 4 several model definitions are used. This section describes these model definitions.

Two types of model definitions where used: model definitions with one rotor and model definitions with three rotors. Only the model definitions with one rotor are featured here as the only difference between the types of model definitions is the number of rotors in the flume.

The model definitions are all similar to Delft3D_1rotor (or Delft3D_3rotor) with some exception: the grid size or use of non-hydrostatic mode or the used layer model or the use of HLES, see Table 18. If HLES is used than partial slip conditions have been used for the walls. The wall roughness length is set to 1.7e-007 m. The used HLES settings can be found in Table 17.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slope in log-log spectrum</td>
<td>1.66666660e+000</td>
</tr>
<tr>
<td>Dimensional number</td>
<td>2</td>
</tr>
<tr>
<td>Prandtl-Schmidt number</td>
<td>0.7</td>
</tr>
<tr>
<td>Spatial low-pass filter coefficient</td>
<td>0.3333333</td>
</tr>
<tr>
<td>Relaxation time</td>
<td>2 min</td>
</tr>
<tr>
<td>Molecular diffusivity</td>
<td>0 m²/s</td>
</tr>
<tr>
<td>Switch to add Elder’s term</td>
<td>Yes</td>
</tr>
</tbody>
</table>

Table 17 HLES parameters.

The vertical and horizontal sub-grid models (such as Delft3D_1rotor_subgrid, Delft3D_1rotor_subgrid_hydrostatic, etc.) have of 15 layers in the z-direction, every layer has a mesh size of 0.015 m (3.125% of depth) except the two layers at the bottom, the middle layer and the top layer. The two bottom layers have a mesh size of 0.0075 m (1.5625% of depth), the middle layer a mesh size of 0.27 m (56.25 % of depth) and the top layer has a grid size of 0.03 m (6.25% of depth).

Note that for depth averaged models it is not possible to use the k-ε model.
<table>
<thead>
<tr>
<th>Model definition</th>
<th>grid size ($\Delta x, \Delta y, \Delta z$) normalized by the rotor diameter D</th>
<th>Non-hydrostatic mode (NH)</th>
<th>Layer model</th>
<th>HLES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delft3D_1rotor</td>
<td>(0.093, 0.093, 0.056)</td>
<td>NH</td>
<td>Z-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_coarse2</td>
<td>(0.185, 0.185, 0.056)</td>
<td>NH</td>
<td>Z-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_coarse2_HLES</td>
<td>(0.185, 0.185, 0.056)</td>
<td>NH</td>
<td>Z-layer</td>
<td>HLES</td>
</tr>
<tr>
<td>Delft3D_1rotor_coarse4</td>
<td>(0.370, 0.370, 0.056)</td>
<td>NH</td>
<td>Z-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_coarse4_hydrostatic</td>
<td>(0.370, 0.370, 0.056)</td>
<td>-</td>
<td>Z-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_coarse4_sigma</td>
<td>(0.370, 0.370, 0.056)</td>
<td>-</td>
<td>Sigma-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_coarse4_HLES</td>
<td>(0.370, 0.370, 0.056)</td>
<td>NH</td>
<td>Z-layer</td>
<td>HLES</td>
</tr>
<tr>
<td>Delft3D_1rotor_Hsubgrid</td>
<td>(3.704, 3.704, 0.056)</td>
<td>NH</td>
<td>Z-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_Hsubgrid_hydrostatic</td>
<td>(3.704, 3.704, 0.056)</td>
<td>-</td>
<td>Z-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_Hsubgrid_sigmalayers</td>
<td>(3.704, 3.704, 0.056)</td>
<td>-</td>
<td>Sigma-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_subgrid</td>
<td>(3.704, 3.704, 1)</td>
<td>NH</td>
<td>Z-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_subgrid_hydrostatic</td>
<td>(3.704, 3.704, 1)</td>
<td>-</td>
<td>Z-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_subgrid_sigmalayers</td>
<td>(3.704, 3.704, 1)</td>
<td>-</td>
<td>Sigma-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_subgrid_HLES</td>
<td>(3.704, 3.704, 1)</td>
<td>NH</td>
<td>Z-layer</td>
<td>HLES</td>
</tr>
<tr>
<td>Delft3D_1rotor_depthav</td>
<td>(3.704, 3.704, 1.67)</td>
<td>NH</td>
<td>Z-layer</td>
<td>-</td>
</tr>
<tr>
<td>Delft3D_1rotor_depthav_HLES</td>
<td>(3.704, 3.704, 1.67)</td>
<td>NH</td>
<td>Z-layer</td>
<td>HLES</td>
</tr>
</tbody>
</table>

Table 18 Model definitions.
H. Model definition with turbulent boundary condition set to zero

This Appendix compares the model definition Delft3D_1rotor_noturb with Delft3D_1rotor. Delft3D_1rotor_noturb has the same characteristics as Delft3D_1rotor except the turbulent boundary conditions at the inflow boundary have been set to zero. The full model details of Delft3D_1rotor can be found in Appendix G.

The ambient turbulence intensity of Delft3D_1rotor_noturb is lower than Delft3D_1rotor. Energy of the ambient flow is transferred to the wake region due to turbulent mixing. Consequently, one can expect that the velocity deficit of Delft3D_1rotor_noturb is maintained longer downstream compared to Delft3D_1rotor and the wake of Delft3D_1rotor_noturb will be narrower.

Figure 87 shows the longitudinal velocity and stream wise turbulent intensity on the centre line of single rotor with data of Delft3D_1rotor and Delft3D_1rotor_noturb. As expected the velocity deficit on the centre line of Delft3D_1rotor_noturb is maintained longer downstream compared to Delft3D_1rotor. Figure 87 shows relative large influence the ambient turbulence has on the recovery of the wake.
Figure 88 Lateral variation of the velocity and stream wise turbulence intensity at sections downstream of a single rotor with data of Delft3D_1rotor and Delft3D_1rotor_noturb.

Figure 89 Vertical variation of velocity and stream wise turbulence intensity at sections downstream of single rotor with data of Delft3D_1rotor and Delft3D_1rotor_noturb.
Figure 88 and Figure 89 show the Lateral and vertical variation of the velocity and stream wise turbulence intensity at sections downstream of a single rotor with data of Delft3D_1rotor and Delft3D_1rotor_noturb. As expected the wake of Delft3D_1rotor_noturb is narrower compared to Delft3D_1rotor. Lower ambient turbulence intensity implies less mixing of the wake flow with the ambient flow.

As stated in Chapter 2 the ambient turbulent intensity is an important parameter for the recovery of the wake. Other studies such as Batten et al. [27] state that it is important to have the correct turbulent intensities around the wake. The comparison of Delft3D_1rotor_noturb with Delft3D_1rotor confirms the importance of the ambient turbulence to accurately predict the evolution of the wake.
I. Momentum loss based on the velocity profile

This appendix elaborates on the estimation of the momentum loss based on the velocity profile.

Introduction

The momentum loss is expressed in the form of a force. The force can be calculated based on the Bernoulli principle, similarly as used in the actuator disc theory, see Appendix A. Multiplication of Equation A.8 results in a force.

\[ p^+ - p^- = \frac{1}{2} \rho (u_0^2 - u_w^2) \]  

Equation A.8

In which P are pressures, \( \rho \) the density and u the velocities, see Appendix A for details. Consequently, the force \( F \) based on the velocity can be calculated by Equation I.1.

\[ dF = \frac{1}{2} \rho (u_0^2 - u^2) dA \]  

Equation I.1

In which \( u_0 \) is the undisturbed velocity of flume without a rotor, \( u \) is the velocity in the wake and \( dA \) the area. Note that differences due to pressures differences in the flume without and flume with rotor are not taken into account.

Estimation of momentum loss of the Delft3D model

By applying Equation I.1 to Delft3D_1rotor and Delft3D_norotor (see Appendix H) the momentum loss based on the velocity can be estimated.

The area \( dA \) of Equation I.1 in this case corresponds to the area of the grid cell. Summation of the contribution of the individual grid cells supplies the total force. If this method is applied at \( x/D = 2 \) downstream of the disc the results yield that the total force \( F = 3.99 \text{ N} \).

Estimation of momentum loss of the experimental data

The momentum loss of the experimental data of Stallard et al. [39] can be estimated by application of Equation I.1 to the velocity profile of the experimental data.

Different to the estimation of the momentum loss based on the Delft3D model, is the fact that not the entire velocity field is known, only the lateral and vertical velocity profile are measured. Consequently, an assumption regarding the velocity field of the experimental data is required: The lateral and vertical measured velocity profile each represent one half of the cross section of the wake. Thus, for example the left part of the measured lateral velocity profile is the same for an area the size of a quarter circle cross section of the wake. Consequently, \( \text{d}A = 0.5 \cdot \pi \cdot r \cdot \text{dr} \) in which \( r \) is the radius and \( \text{dr} \) is the distance of the measurements. Summation of all the measurements contributions gives the total force. If this method is applied at \( x/D = 2 \) downstream of the rotor the results yield that the total force \( F = 5.35 \text{ N} \).
J. Supplementary Figures
This appendix shows supplementary figures of the report.

*Self-similar solution defined by the conditions set by the experimental data*

Figure 90 Velocity downstream of the axis of a single rotor, with data of Stallard et al. [39] and the self-similar solution of the axi-symmetrical wake defined from several distances downstream by the measurement data. The error bars indicate 15% deviation from the velocity.

Figure 91 Velocity downstream of the axis of a single rotor, with data of Stallard et al. [39] and the self-similar solution of the planar wake defined from several distances downstream by the measurement data. The error bars indicate 15% deviation from the velocity.

The lateral velocity of the self-similar solutions requires the parameter δ to be calibrated for the considered wake. In this case this has been done on the single rotor of the measurements of Stallard et al. [39] at x/D = 2 downstream of the rotor, see Figure 92.
Figure 92 Lateral velocity at $x/D = 2$ downstream of the rotor. With data of Stallard et al. [39] (Blue) and self-similar solution (red) defined by the data of Stallard et al. [39].

Figure 93 Lateral variation of the velocity downstream of a row of three rotors at 1.5D lateral spacing, with data of Stallard et al. [39] (blue), self-similar solution of axi-symmetrical wake (A, magenta) and self-similarity solution of planar wake (B, cyan).
Lateral and vertical variation of the velocity and streamwise turbulence intensity of Delft3D_1rotor

Figure 94 Lateral variation of the velocity and streamwise turbulent intensity for a single rotor at x/D = 2 downstream of rotor, with data of Stallard et al. [39], Delft3D_1rotor and self-similarity solution of the planar wake.

Figure 95 Vertical variation of the velocity and streamwise turbulence intensity of single turbine at x/D = 2 downstream of the rotor, with data of Stallard et al. [39], Delft3D_1rotor and self-similarity solution of the planar wake.
**Vertical velocity and stream wise turbulence intensities of HLES models**

Figure 96: Vertical variation of A) velocity and B) stream wise turbulence intensity downstream of the middle rotor of a row of three rotors, with data of Stallard et al. [39] and various Delft3D HLES grid model definitions with varying grid sizes. Note the differences in scale.
Lateral and vertical velocity and stream wise turbulence intensities of hydrostatic models

Figure 97 Lateral variation of A) velocity and B) stream wise turbulence intensity downstream of a row of three rotors at 1.5D lateral spacing. Data from Stallard et al. [39], Delft3D_3rotor and Delft3D_3rotor_hydrostatic. The highlighted area in blue indicates deviation of 15% of the data from Stallard et al. [39]. Note the differences in scale.
Figure 98 Vertical variation of A) velocity and B) stream wise turbulence intensity downstream middle rotor of a row of three rotors. Data from Stallard et al. [39], Delft3D_3rotor and Delft3D_3rotor_hydrostatic. Note the differences in scale.
Figure 99 Lateral variation of A) velocity and B) stream wise turbulence intensity downstream of a row of three rotors at 1.5D lateral spacing. With data of Stallard et al. [39], Delft3D_3rotor_Hsubgrid, Delft3D_3rotor_Hsubgrid_hydrostatic, Delft3D_3rotor_subgrid and Delft3D_3rotor_subgrid_hydrostatic. The highlighted area in blue indicates deviation of 15% of the data from Stallard et al. [39]. Note the differences in scale.
Figure 100 Vertical variation of A) velocity and B) stream wise turbulence intensity downstream middle rotor of a row of three rotors. With data of Stallard et al. [39], Delft3D_3rotor_Hsubgrid, Delft3D_3rotor_Hsubgrid_hydrostatic, Delft3D_3rotor_subgrid and Delft3D_3rotor_subgrid_hydrostatic.

Note the differences in scale.
Lateral velocity and stream wise turbulence intensities of models with varying grid layer model

Figure 101 Lateral variation of A) velocity and B) stream wise turbulence intensity downstream of a row of three rotors at 1.5D lateral spacing. Data of Stallard et al. [39], Delft3D_3rotor_Hsubgrid_hydrostatic, Delft3D_3rotor_Hsubgrid_sigma, Delft3D_3rotor_subgrid_hydrostatic and Delft3D_3rotor_subgrid_sigma. The highlighted area in blue indicates deviation of 15% of the data from Stallard et al. [39].
**Velocity difference and water level at Marsdiep case study**

**Figure 102** Absolute velocity difference in m/s in the middle of the water column at spring low water level. Values smaller than 0.01 m/s not shown.

**Figure 103** Water level difference in m during low water spring tide. Values in between -0.001 and 0.001 m not shown.
K. HLES and sigma-layer model

This appendix investigates the influence of HLES in combination with the sigma layer model.

![Graph showing velocity and stream wise turbulence intensity](image)

Figure 104 shows the Figure 105 Velocity and stream wise turbulence intensity downstream of the axis of a single rotor with data of Stallard et al. [39], Delft3D_1rotor_hydrostatic, Delft3D_1rotor_hydrostatic_sigma and Delft3D_1rotor_hydrostatic_sigma_HLES. The model definitions are similar to Delft3D_1rotor (see Appendix G) with the exception of the use of hydrostatic mode, sigma-layer model or the use of HLES. It can be seen that the use of HLES in combination with the sigma-layer model lead to higher stream wise turbulence intensities, similar as for the use of HLES in combination with the Z-layer model (see Paragraph 4.5). However, it appears that the use of HLES in combination with the sigma layer model produces more extra stream wise turbulence intensities compared to the extra stream wise turbulence intensities produced by HLES in combination with the Z-layer model. The influence of HLES in combination with the sigma-model is small. However the influence on the velocities seem slightly large compared to the use of HLES with the Z-layer model.

The influence of HLES in combination with the sigma-model leads to higher turbulence intensities compared to the use of HLES in combination with the Z-layer model. However, the influence on the velocities are small.
This page has been left blank on purpose