Proceedings of the ASME 2015 34th International Conference on Ocean, Offshore and Arctic Engineering
OMAE2015
May 31-June 5, 2015, St. John's, Newfoundland, Canada

OMAE2015-41187

CFD STUDY ON FREE-SURFACE INFLUENCE ON TIDAL TURBINES IN HYDRAULIC STRUCTURES

Aldo Tralli
Deltares
Delft, Netherlands
Email: aldo.tralli@deltares.nl

Arnout C. Bijlsma
Deltares
Delft, Netherlands
Email: arnout.bijlsma@deltares.nl

Wilbert te Velde
Tocardo
Den Oever, Netherlands
Email: wtv@tocardo.com

Pieter de Haas
Tocardo
Den Oever, Netherlands
Email: pdh@tocardo.com

ABSTRACT

In order to estimate the impact on energy production and environment of tidal turbines placed in or near hydraulic structures like discharge sluices or storm surge barriers, a Computational Fluid Dynamics (CFD) study has been carried out on the relation between (head) loss induced by the turbines and their gross power production.

CFD computations have been performed for Tocardo T2 turbines, using STAR-CCM+.

Simulations of a single turbine in free flow conditions compare favorably with results of Blade Element Momentum (BEM) computations, in terms of torque and thrust. This BEM method model had been previously validated against both CFD data and field measurements.

Then, a series of tests has been performed in a "virtual tow tank", including the effect of the free surface and the blockage by side and bottom walls.

These computations provide a base for a first estimate of the effect of turbines on the discharge capacity of a generic structure. This is considered to be the first step in a more general approach in which ultimately the effect of tidal turbines in the Eastern Scheldt Storm Surge Barrier will be assessed.

INTRODUCTION

Computational Fluid Dynamics (CFD) has established itself as a valuable and reliable tool in the investigation and design of applications involving flow through rotating equipment, thanks to the wealth of validation data built up over time. Tidal turbines are a relatively small and new niche in the realm of rotating machinery, but even for them the amount of existing literature is quite abundant. Advanced CDF codes allow to model the real-time rotation of the turbine blades and hub in a transient mode by defining an inner sub-domain with a rotating or sliding mesh, see e.g. [1–3]. This approach resolves the full interaction between rotor blades and the fluid, and potentially provides the highest accuracy when the flow field in the vicinity of the turbine needs to be represented in detail, e.g. when the fluid flow through the turbine is perturbed by external features, such as the turbine strut, other support structures, the bathymetry, or the adjacent turbines. However, the interaction with the free surface was not included in [1–3].

Computationally more efficient is the concept of a 'frozen rotor'. This can be applied for problems in which the flow through the turbine can be considered steady state, see e.g. [4]. A third option consists of a mixed RANS-BEM approach, see [5]. By lack of suitable full scale measurements, several studies validate their CFD modelling approach to scale model test. However, [3] concludes that considerable scale effects can be expected, making validation on scale model tests not very attractive.

The objective of this study is to leverage the confidence in CFD as a valid design tool in tidal energy applications, to gain further insight in the interaction between an operating turbine and the surrounding environment, namely the free surface (between water and air) and possibly the sea bottom and the surrounding structures. More specifically, the interest is in finding metrics and performance indicators for the whole system.

The final aim is to optimize the placement of the turbine(s), in order to exploit said interactions and increase the energy yield. This study has been carried out in the framework of the design of an array of tidal turbines in the Eastern Scheldt Storm Surge Barrier, therefore the envelope of possible flow conditions roughly matches said installation.

The array exists of five Tocardo T2 bi-directional horizontal axis tidal turbines, with a rated power output of 1 MW (4 m/s). The fixed pitch twin blade turbines consist of a nacelle of 1.3 m diameter and a rotor of ca. 5 m diameter. The blade angle automatically changes when the flow reverses. A shaft connects the nacelle to the support structure above the water. This type of turbine will be used in the present study.

Moreover, the CFD model is also meant to be used as a nested entity within the large-scale open source Hydrodynamic modeling code Delft3D-FLOW (see: oss.deltares.nl): at this stage, the results of the CFD model shall be used to set up a "black box" to be implemented in Delft3D-FLOW, to mimic the behavior of the turbine.

This objective shall be achieved in three steps: (1) Validation of the CFD model against proven Blade Element Momentum (BEM) results, (2) Study of the interaction between the free surface and the turbine performance in a model problem, and definition of the performance indicators, and (3) Study of the interaction between turbine, free surface and a realistic approximation of the bathymetry in the vicinity of the Eastern Scheldt Storm Surge Barrier (this latter stage is not covered in this paper).

METHODOLOGIES

Blade Element Momentum Method

The BEM Method applies Blade Element Theory and Axial Momentum Theory to calculate the forces on a wind turbine rotor [6]. Blade Element Theory divides blades into sections along the span that are assumed independent of each other. Axial Momentum Theory applies the conservation laws on a 1D stream tube in axial direction. From the momentum lost in the flow the induced velocities in the axial and tangential directions are derived. These induced velocities are used to calculate the lift and drag forces on each blade element, from which in turn the axial thrust and the tangential forces are calculated. This process is repeated until both methods produce equal results. Blade Element Theory assumes steady state and two dimensional homogeneous axial flow, without surface, bottom and side wall interaction. In practice pressure variations along the span result in radial flow. To compensate for these shortcomings corrections are applied to the original theory.

The BEM software package used to compute the forces on the T2 turbine is the Blade Optimisation Tool Hydro (BOT Hydro) by the Netherlands Energy Research Centre (ECN), which includes corrections for tip and root losses (Prandtl), turbulent wake state and a 3D correction based on the work of Snel *et al*. for rotational effects, see [7–9]. Using an initial design, BOT can be used to optimize the supplied chord and twist distribution along the blade to obtain maximum annual energy yield for a given water speed distribution. The aerodynamic coefficients of the air foils in BOT Hydro were generated using RFoil, a modification of XFoil by Drela [10]. Using BOT Hydro the dimensionless curves for power, torque and thrust (c_P-λ, c_Q- λ and c_T- λ curves) of the T2 turbine were produced.

Numerical setup of CFD

This section provides specific considerations on the physics and geometry modeling required for the CFD analysis. The commercial solver Star-CCM+ has been used. The flow through the turbine is assumed to be governed by the Reynolds-averaged Navier-Stokes (RANS) equations. The equations are discretized by means of a Finite Volume scheme on an unstructured hexahedral grid. The k-ω shear stress transport model is used as a turbulence model for viscous flows.

A Tocardo T2 turbine has been simulated, based on a CAD drawing of the two-bladed rotor, with a diameter D of approximately 5 m, connected to its nacelle. The strut connecting the turbine with the support structure is also present.

The computational domain is a prismatic channel: the inflow and outflow sections are square, with a side length of 4D. The inlet section is located 4D upstream of the turbine, and the outlet is 10D downstream, see Fig. 1. Only one flow direction has been evaluated: the incoming flow on the nose.

The size of the computational domain is chosen to roughly match the size of one single grid element of a typical Delft3D-FLOW computation.

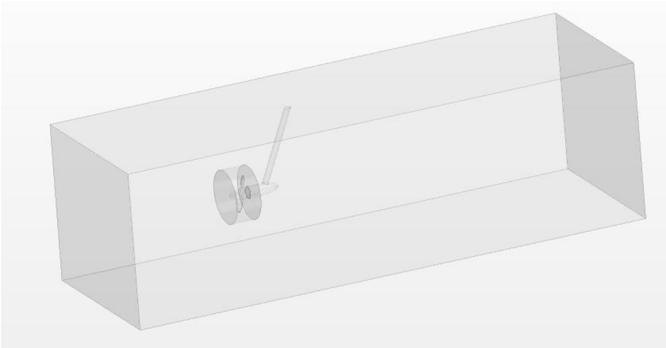

Figure 1. REPRESENTATION OF THE SIMULATION DOMAIN: THE BLADES ARE MODELLED IN DETAIL, AND THE ROTATING DOMAIN IS VISIBLE.

When defining the simulation strategy, a choice is to be made between a real-time rotation of the turbine blades (involving the displacement of the mesh nodes in an unsteady setting), and an approach commonly known as frozen rotor whereby an unsteady motion problem is converted to a steady-state problem by imposing a moving frame of reference on a static mesh. Upon verification that the quantitative values of the most important characteristics (torque and thrust at the turbine) were very close in the two approaches, it has been decided to follow the cheapest approach from computational standpoint: the results in this study were obtained by a frozen rotor model.

CFD computations

Using the numerical set-up described above, we first validated the CFD modeling approach against BEM results available for the turbine in question. Also a sensitivity study has been performed, to ensure that the flow and its most important characteristics (torque and thrust of the turbine) were independent of the discretization both in space and time, to estimate the uncertainty related to the grid size.

After a successful validation, the interaction between the free surface and the turbine is investigated. More specifically, the effect of the proximity of the free surface on the turbine torque, and the (additional) head losses induced by the turbine are investigated.

The relatively small size of the computational domain is expected to affect the simulation results in a number of ways: it will produce effects on the computed thrust and torque, to an extent which albeit small, has not been quantified in this study. It will also lead to a small underestimation of the head loss, since the wake will not be fully recovered before the outlet is reached. The reason for the selection of a rather small domain lies in the desire of using the same, validated computational grid also for the study on the turbine-free surface interaction. In that case, a small cross section would produce a larger (and easier to measure) difference in water level.

RESULTS AND DISCUSSION

CFD validation

A preliminary step has been the validation of the simulation choices against BEM data provided by Tocardo, which were deemed to be accurate, due to previous validation of the modeling approach, performed using experimental data from other turbines [11].

A single-phase simulation has been set up, using sea water ($\rho = 1023\ kg/m^3$) as working fluid. Water velocity was assigned at the inlet, and static pressure was assigned to zero (i.e. the reference pressure level) at the outlet and on the side walls of the prismatic channel. Gravity was not included in the model at this stage.

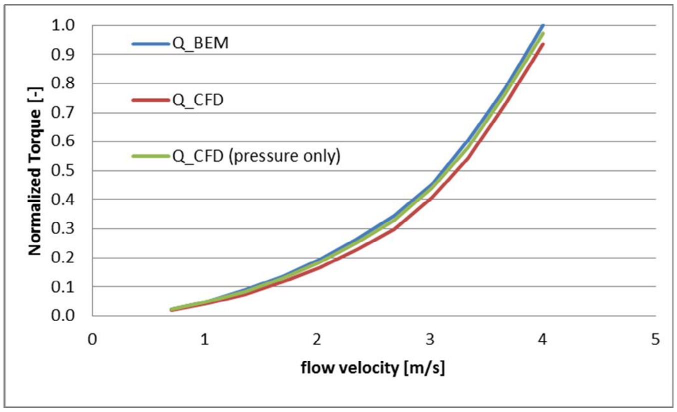

Figure 2. TORQUE COMPUTED BY BEM AND CFD AS A FUNCTION OF THE FREE STREAM VELOCITY.

This simulation set up was used to verify that the modeling assumptions, the grid sizing and time step were actually fit for the purpose.

A full sweep of the operating condition was simulated, with changing upstream velocity and matching rotational velocity of the turbine.

The results are summarized in Fig. 2, where the normalized torque as a function of the free stream water velocity is displayed, and comparison is made between the BEM data and the CFD.

In this article, normalized values are obtained by dividing the torque by the rated value (i.e. the BEM torque at a free-stream velocity of 4 m/s).

Two curves were taken from the CFD computation, displaying the torque computed integrating the whole stress tensor, or accounting only for the pressure (and discarding the shear stress).

The torque obtained via CFD appears to be about 12% lower than the BEM values, as an average over all measurement points: a possible explanation of the discrepancy might be due to the fact that the BEM assumes inviscid fluid, while the CFD calculation is performed on a viscous one. Calculating the CFD torque only based on the pressure (thus discarding the shear components of the stress tensor), produces a torque 4% lower than the BEM values.

Figure 3 shows a similar comparison for the normalized thrust (i.e. divided by the BEM-calculated thrust at 4 m/s free stream velocity): in this case, the contribution of the shear stress to the thrust is negligible, compared to pressure. The CFD thrust is about 7% lower (as an average over all measurement points) than that computed via BEM.

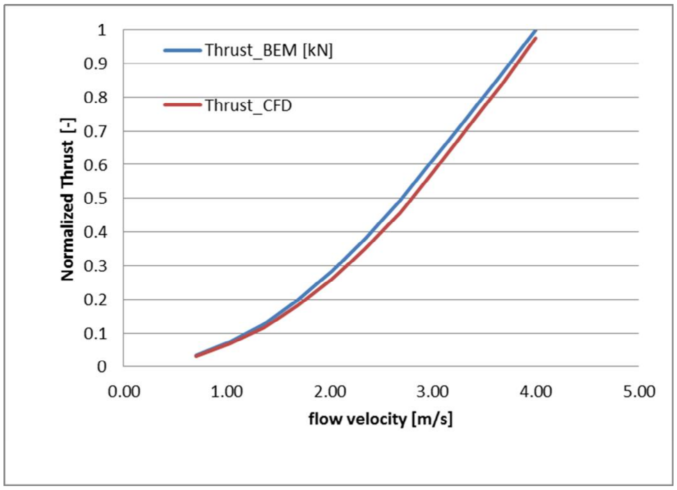

Figure 3. THRUST COMPUTED BY BEM AND CFD AS A FUNCTION OF THE FREE STREAM VELOCITY.

Interaction between the free surface and the turbine

In order to relate the energy collection to the presence and position of the free surface, a "virtual tow tank experiment" has been set up: a two-phase fluid model, including sea water ($\rho = 1023\ kg/m^3$) and air (incompressible gas at 20ºC and 1 Atm) is considered with the implementation of a Volume of Fluid (VOF) algorithm, for accurate modeling of the free surface flow. A uniform velocity vector, and reference water level are assigned at the upstream boundary, while a hydrostatic pressure profile is assigned on the downstream boundary, matching the free surface level in undisturbed conditions.

The side and bottom boundaries are assigned to be free-slip walls: this will ensure that a known amount of water will flow through the whole system, and the flow resistance is entirely induced by the turbine and attached structures (since no boundary layer will develop on the side walls). Atmospheric pressure is assigned on the top boundary.

The size of the domain and the mesh are the same as described during the validation case. The purpose of this set up is twofold: control the blockage effect on the flow, limiting it to the contribution of the turbine, and ensuring that a known amount of water flows through the whole domain.

Two-phase simulations display a stronger transient behavior, due to the free surface and its interaction with the strut and the rotor: a pattern of free surface waves tends to form at the strut, propagate in the field with reflections at the walls, see Fig.4. It has therefore been elected to perform those simulations in transient mode.

The transient behavior has very little effect on the main flow parameters (torque, thrust, dissipated power). However, the amplitude of the free surface waves appears to be of the same order of magnitude as the head loss. As a consequence, long transients need to be simulated, in order to effectively filter out the waves from the head.

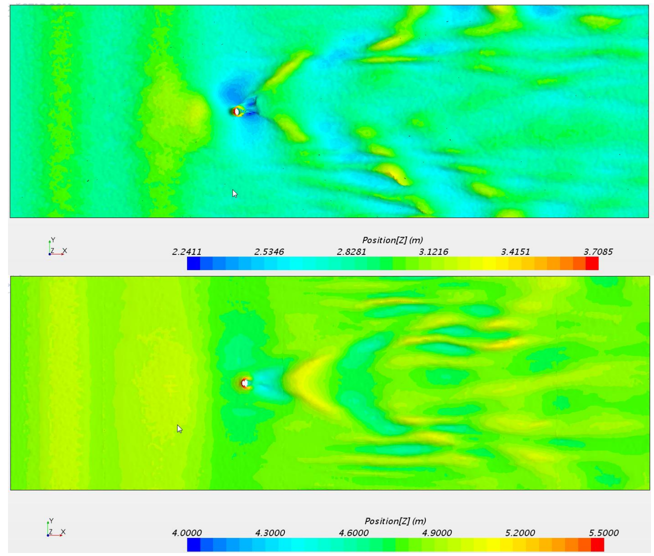

Figure 4. COMPARISON OF THE FREE SURFACE SHAPES. ABOVE: AXIS DEPTH 0.55D, BELOW: AXIS DEPTH 0.9D.

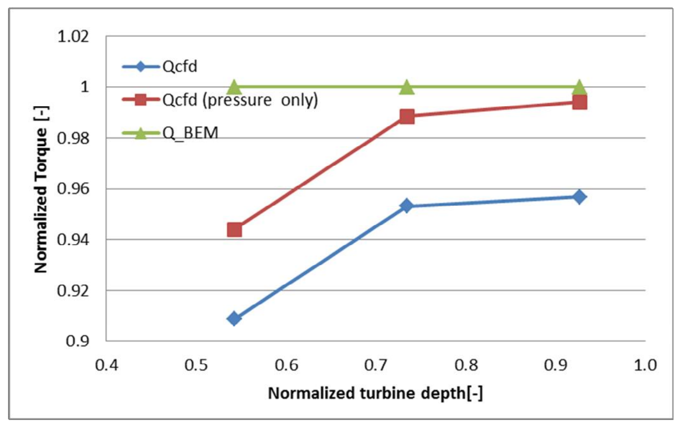

Figure 5. NORMALIZED TORQUE AS A FUNCTION OF NORMALIZED DEPTH OF THE TURBINE AXIS.

Two checks are performed: the effect of the free surface on the amount of generated torque, and the head loss induced on the flow by the turbine: the nominal power drawn from the fluid by the turbine translates in a head between the portion of tow tank upstream of the turbine, and the portion downstream.

Effect of the proximity of the free surface on the turbine torque

By design, the rotation axis of the turbine is supposed to be placed about 1D below the "nominal" water surface. A series of three simulations is performed, studying the effect of decreasing such distance on the measured torque. The results are shown in Fig. 5, where the torque computed in the CFD model, is compared to the "nominal" torque, evaluated with the BEM technique for a free stream velocity of $V_{upstream}$ = 4 m/s. As before, two values for the torque are provided, with and without the shear contribution. The CDF computation shows that for very low water levels, where the axis depth is approx. 0.5D, the torque, and thus the power output of the turbine reduces by almost 10%.

Estimate of the head losses induced by the turbine

The losses induced by the turbine are measured in this study in terms of energy head:

$$\Delta H = \frac{\overline{p}_{tot\_upstream}}{\rho g} - \frac{\overline{p}_{tot\_downstream}}{\rho g} \tag{1}$$

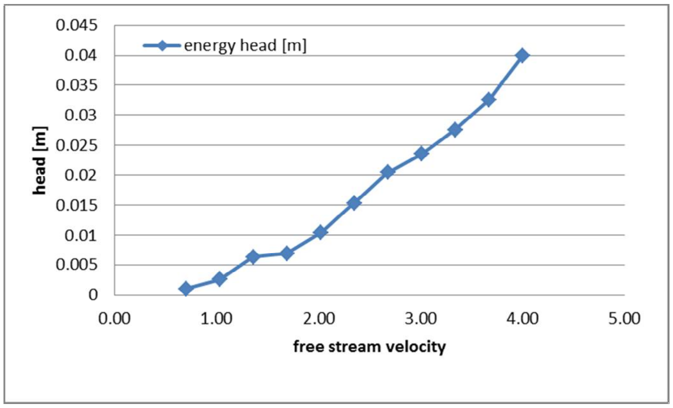

Figure 6. COMPUTED HEAD LOSS OVER THE TOW TANK WITH TURBINE, AS A FUNCTION OF THE FREE STREAM VELOCITY. AXIS DEPTH 0.9D.

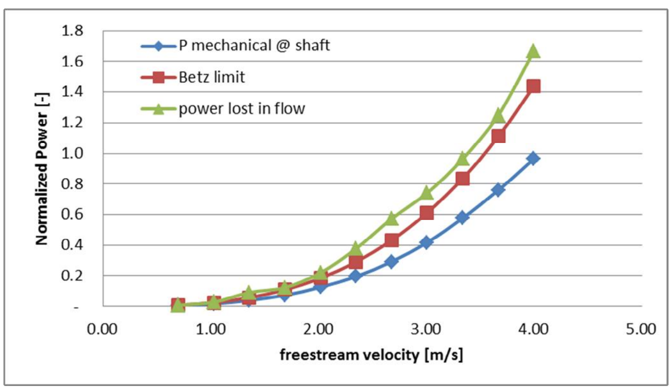

Figure 7. COMPARISON BETWEEN THE POWER SUBTRACTED FROM THE FLOW, AND THE POWER COLLECTED AT THE TURBINE SHAFT. AXIS DEPTH 0.9D.

where $\overline{p}_{tot\_upstream}$ and $\overline{p}_{tot\_downstream}$ are the are the space-average values, computed over the cross-sectional area, of the total pressure at the inlet and outlet of the domain, respectively, g the acceleration due to gravity and ρ the water density. The total pressure is defined as:

$$p_{tot} = p + \frac{1}{2}\rho V^2 \tag{2}$$

where p is the static pressure and $\frac{1}{2}\rho V^2$ the dynamic pressure. The resulting head losses as a function of the free stream velocity are presented for the 0.9D axis depth case in Fig. 6.

Based on the computation of the head loss through the virtual tow tank, it appears that the total losses in the flow amount to about 170% of the power drawn by the turbine, see Fig. 7.

It's worth noting that, according to the simulation, the energy collected by the turbine is about 68% of the theoretical maximum, calculated via the Betz formula.

$$P_{Betz} = \frac{16}{27}\frac{1}{2}\rho A_{disk} V_{upstream}^3 \tag{3}$$

A_{disk} in Eqn (3) is the area swept by the rotor blades and $V_{upstream}$ the mean velocity at the inlet of the domain.

Ultimately, in the Eastern Scheldt Storm Surge Barrier five turbines will be placed in a 40 m wide gate with a sill depth of 9.5 m, which corresponds to a smaller domain than the one investigated (about 1.5D wide and 1.8D deep for a single turbine). Due to this the blockage ratio would increase from about 5% to some 30%, improving the efficiency of the turbine and increasing the head loss.

CONCLUSIONS AND FURTHER WORK

The current set of simulation has provided an initial benchmark for the validation of a CFD model studying the performance of a tidal turbine in the proximity of the free surface: shortening the distance between the free surface and the turbine axis has a strong negative effect on the turbine performance.

Moreover, it appears to be possible to establish a relation between the power collected by the turbine, and the head loss in the flow.

The next step is to investigate the effects on thrust, power and head loss of narrowing and/or depth reduction of the channel domain by CFD. Using the CFD results a blockage correction factor can be determined and compared with the blockage correction factor with respect to the infinite domain based on actuator disk theory, see [4].

In the near future, the CFD approach will be repeated in a more realistic setting, including a number of turbines, positioned in a hydraulic structure and with a realistic bathymetry. The resulting data, coupled with a far-field hydrodynamic simulation tool such as Delft3D-FLOW, will allow us to study the interaction between said components, possibly optimizing the placement of the turbines, and quantify a number of phenomena of interest in the areas affected by tidal turbines: tidal dynamics and sediment transport, the volume of water exchanged through the hydraulic structure, water quality and ecology.

ACKNOWLEDGMENTS

The CFD work was carried out in the Energising Deltas project of the Program ‘Opportunities for West’, funded by the European Regional Development Fund (ERDF) and the Province of Noord-Holland. Tocardo International BV is gratefully acknowledged for the permission to publish the results of the BEM and CFD computations.

REFERENCES

[1] Lawson, M., Li, Y., and Sale, D., 2011. “Development and verification of a computational fluid dynamics model of a horizontal-axis tidal current turbine”. In Proceedings of the 30th International Conference on Ocean, Offshore, and Arctic Engineering. Paper OMAE2011-49863.

[2] Kolekar, N. and Banerjee, A. (2013). A coupled hydro-structural design optimization for hydrokinetic turbines. Journal of Renewable and Sustainable Energy 5, 053146 (2013); doi: 10.1063/1.4826882.

[3] Otto, W., Rijpkema, D., and Vaz, G. (2012). Viscous-Flow Calculations on an Axial Marine Current Turbine. Proceedings of the 31st International Conference on Ocean, Offshore and Arctic Engineering. Paper OMAE2012-83452.

[4] Jo, C.H., Yim, J.Y., Lee, K.H., and Rho, Y.H. (2012). Performance of horizontal axis tidal current turbine by blade configuration. Renew Energ, 42, pp. 195–206.

[5] Turnock, S., Phillips, A., Banks, J., Nicholls-Lee, R. (2011). Modelling tidal current turbine wakes using a coupled RANS-BEMT approach as a tool for analysing power capture of arrays of turbines. Ocean Engineering, 38 1300–1307.

[6] Glauert, H. (1935). Airplane propellers. In Aerodynamics Theory, 169-360. Durand W.F. (ed) Dover New York.

[7] Bot, E.T.G. and Ceyhan, O. (2011). Blade Optimization Tool BOT. User Manual. ECN-E--09-092. Petten, The Netherlands.

[8] Schepers, J.G. (2012). Engineering models in wind energy aerodynamics - Development, implementation and analysis using dedicated aerodynamic measurements. PhD Thesis, Technical University of Delft.

[9] Snel, H., Houwink, R. and Bosschers, J. (1994). Sectional prediction of lift coefficients on rotating wind turbine blades in stall. ECN-C--93-052, Petten, The Netherlands,

[10] Drela M. (1989). XFOIL: An analysis and design system for low Reynolds number airfoils. In Low Reynolds number aerodynamics, Lecture notes in Engineering 54, 1–12.

[11] Jacobs, V. (2014). Validation of BEM-T and RANS Simulations with Experiments on Tidal Turbines. Master Thesis TU Delft.