Development of an Adapted Turbine Model for Hydrokinetic Turbines in 2d Shallow Water Solvers

Christa Stadler, Jonas Wack, Stefan Fraas and Stefan Riedelbauch
Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70569 Stuttgart, DE
E-mail: christa.stadler@ihs.uni-stuttgart.de

Abstract.
Identifying optimum sites for hydrokinetic turbines in rivers and oceans in order to gain a maximum power output is more and more focused through the demand for sustainable energy production. This task may be supported by 2d shallow water solvers, which are able to simulate the flow in large areas with comparatively low effort. The main problem using them for turbine siting is that those solvers work with depth averaged flow quantities. Therefore, it is not possible to directly resolve the geometry of a kinetic turbine. A mathematical model is required representing the turbine characteristics in the flow field. Existing turbine models are usually based on the Linear Momentum Actuator Disk Theory which was derived for an ideal propeller in an infinitely large flow field. Regarding rivers and shallow or narrow tidal channels but also more complex types of hydrokinetic turbines - like diffuser augmented turbines - those assumptions are not fulfilled. This work presents the development of a turbine model adapted to those requirements and its implementation in 2d shallow water solvers.

1. Introduction
Driven by the transition towards sustainable energy production alternative technologies, like hydrokinetic turbines, receive increased attention. Those turbines are particularly suitable for ecological sensible or remote areas. In order to gain maximum power output through an optimum positioning of the turbines in potential project areas CFD simulations are a powerful tool. As project areas are typically of large size, 2d depth averaged shallow water solvers are commonly used for this purpose. In a two dimensional solution domain a three dimensional turbine geometry may not be resolved. To be able to correctly determine the turbine power a 2d turbine model describing the characteristics of the actual hydrokinetic turbine is required. For common horizontal-axis propeller turbines at locations with large flow cross section the Linear Momentum Actuator Disk Theory (LMADT) introduced by Lanchester and Betz may be used [1]. For installation at very shallow water sites the turbine rotor is typically diffuser augmented (DAT) to increase the effective cross section [2][3]. Those more complex turbine designs may not be described directly by the LMADT. Also the installation in small flow cross sections (e.g. rives) makes a modification of the LMADT necessary, as it is only valid for an infinitely large flow field around the propeller [4]. This work presents the development of a turbine model for 2d shallow water solvers satisfying the described requirements.
2. The basic turbine model

The adapted turbine model presented in this work is based on the turbine model described in [5] and [6]. A similar approach implemented in a different shallow water solver is discussed in [7]. The main modelling concept is to increase the drag in one or more cells at the specified turbine site. This procedure ensures that the influence of the turbine on the flow and vice versa is taken into account and a realistic power output may be computed by the 2d solver. The specific value of the corresponding drag coefficient $c_d,t$ is calculated determining the rotor thrust according to the LMADT. Therefore, the induction factor $a$ is introduced describing the deceleration of the flow caused by the rotor.

$$a = 1 - \frac{u_r}{u_\infty}$$ (1)

Therein, $u_\infty$ denotes the undisturbed upstream velocity and $u_r$ is the transport velocity in the rotor plane. The model furthermore assumes that the undisturbed velocity profile is decelerated over the height of the turbine $h_r$, whereby the rest of the profile is not affected by the rotor. The principle is illustrated in Figure 1. Integrating the velocity profile disturbed by the rotor, the following relation between the mean velocity in the turbine cell $\bar{u}$ (disturbed) and the upstream velocity $u_\infty$ (undisturbed) is derived:

$$\bar{u} = \frac{h - h_r}{h} u_\infty + \frac{h_r}{h} \left(1 - a\right) u_\infty = u_\infty \left(1 - \frac{h_r}{h} a\right)$$ (2)

It should be noted that the explained procedure is originally applied for a 1/7-power law velocity profile [5]. In this work, a block profile with constant velocity over the flow depth is used. This makes the implementation and validation of the modifications described in the following easier and is - at this stage of the work - not relevant for the modelling concept. However, knowing the upstream undisturbed velocity the rotor thrust may be calculated according to the LMADT:

$$T_{AD,3d} = \frac{\rho}{2} A_r c_T u_\infty^2 = \frac{\rho}{2} A_r 4a \left(1 - a\right) u_\infty^2$$ (3)

To achieve the correct induction factor in the 2d shallow water model this 3d rotor thrust must be equal to the 2d thrust in the turbine cell. Based on this assumption and using Equation 3 the two dimensional drag coefficient $c_{d,t}$ is derived:

$$c_{d,t} = \frac{A_r 4a \left(1 - a\right) u_\infty^2}{2 A_{cell} a^2}$$ (4)

The drag in the turbine cell is increased by this value in order to model the kinetic turbine. The turbine power may be post processed according to the LMADT. This model is based on the assumption that the rotor faces ideal flow conditions in terms of inflow direction.

3. The adapted turbine model

A modification of the basic turbine model introduced above is necessary as it uses the LMADT which is only valid for an ideal single rotor in an infinitely large flow field. The adapted turbine model has to be able to describe DATs and/or turbine operation in small channel cross sections. Finally, the new model has to be implemented in the shallow water solver.

3.1. Approach for diffuser augmented turbines

At first, an approach to model DATs with or without slots is developed to obtain a 1d mathematical description for those machines. As described in [3] and [8] an analytical or semi-empirical model is only feasible for simple diffuser geometry. More complex turbine geometries
often feature slots and spoilers. Therefore, it is not possible to apply the analytical or semi-empirical models from literature here. A simple empirical model is developed using several characteristic values gained from numerical 3d flow simulations of full turbine geometries in an infinite flow field. Those characteristic values are defined in Table 1.

**Table 1. Definitions of empirical values for modelling DATs.**

| pressure coefficient | induction factor | thrust ratio | massflow ratio |
|----------------------|------------------|--------------|----------------|
| \( c_{p,DAT} \) = \( \frac{u_r A_r \Delta p_r}{\frac{1}{2} \rho A_d u_\infty^3} \) | \( a = 1 - \frac{u_d}{u_\infty} \) | \( \tau = \frac{T_r}{T_d + T_r} \) | \( q = \frac{u_d A_d}{u_r A_r} \) |

For the development of the DAT-model, the LMADT is adapted as follows: The projected diffuser outlet area \( A_d \) is used as reference plane instead of the projected rotor area \( A_r \). The maximum theoretical power corresponds to a rotor with the size of \( A_d \). This is a common approach in literature [2]. The power output of the machine may then be formulated as follows:

\[
P_{DAT} = \frac{1}{2} \rho A_d u_\infty^3 c_{p,DAT}
\] (5)

Applying the continuity equation, the transport velocity in the rotor plane \( u_r \) may be calculated using the induction factor of the whole machine \( a \), which is determined according to Table 1.

\[
u_r = \frac{u_d A_d}{q A_r} = \frac{(1 - a) u_\infty A_d}{q A_r}
\] (6)

For a slotted or multistage diffuser the massflow ratio \( q \) has to be considered. It describes the relation between rotor discharge and the discharge through the diffuser outlet, which is the sum of the rotor discharge and the discharge through the slots. For a single stage diffuser without slots \( q \) becomes one. From Equation 5 and 6 the rotor thrust is calculated as:

\[
T_r = \frac{P_{DAT}}{u_r} = \frac{\rho q}{2(1 - a)} A_r c_{p,DAT} u_\infty^2
\] (7)

For DATs the total thrust on the machine is shared by the thrust on the rotor and the thrust on the diffuser [8]. Consequently, the force acting on the flow, which is required for the correct
definition of the drag coefficient in the shallow water solver, is the sum of those forces. Using the thrust ratio \( \tau \) this force may be expressed as:

\[
T_{\text{DAT},3d} = \frac{T_r}{\tau} = \frac{\rho q}{2(1-a)\tau} A_r c_{p,DAT} u_\infty^2
\]  

(8)

As mentioned previously, the quantities \( c_{p,DAT}, a, q \) and \( \tau \) have to be determined through 3d CFD-simulations. The determination of the undisturbed upstream velocity \( u_\infty \) is not affected by the adaption of the basic model and may be applied according to Equation 2 when replacing the diameter of the rotor \( h_r \) by the height of the projected diffuser area \( h_d \).

### 3.2. Hydrokinetic turbines in channel flow

Placing kinetic turbines in narrow or shallow flow cross sections has a significant influence on the maximum power output. This phenomenon is analytically investigated and discussed in detail by [4]. According to their work the maximum undisturbed pressure coefficient \( c_{p,\infty} \) for an ideal kinetic turbine is influenced by the limiting channel boundaries as follows:

\[
c_{p,c} = \frac{c_{p,\infty}}{(1-\epsilon)^2}
\]  

(9)

Also the corresponding induction factor \( a_c \) is influenced:

\[
a_c = \frac{a_\infty + \epsilon}{1 + \epsilon}
\]  

(10)

Therein, \( \epsilon \) is the blockage ratio, which is defined as the ratio of the projected rotor area \( A_r \) and the channel cross sections \( A_c \). This correction of \( c_{p,c} \) and \( a_c \) is derived from 1d momentum, energy and mass conservation and is not directly applicable to channels with small aspect ratios \( \alpha = \frac{w}{h} \) [9]. As large rivers with limited flow depth are a potential project area for DATs the influence of the aspect ratio is investigated in detail through a series of 3d numerical flow simulations using the openFOAM solver simpleFoam (foam-extend-3.1). A completely flat actuator disk (no axial extension) inducing a constant pressure loss is placed in rectangular channels with varying blockage and aspect ratios.

![Figure 3. Influence on AD in different channel geometries - pressure coefficient (left) and induction factor (right)](image)

Regarding the results presented in Figure 3 it may be noticed that laminar simulations agree quite well with the theoretical values of \( c_{p,c} \) and \( a_c \) for a quadratic channel cross section. A change in aspect ratio has only very little influence on the results. With increasing \( \epsilon \) this influence even tends to zero.
The results are remarkably changing for turbulent flow simulations using the $k$-$\epsilon$-model. The deviation from the laminar simulation results in $c_{p,c}$ and $a_c$ is largest for small blockage ratios and decreases with increasing $\epsilon$. This may also be observed looking at the results presented in [9]. The results lead to the conclusion that turbulent effects become less important in heavily blocked channels. Furthermore, the influence of the aspect ratio on the solution is more developed regarding the turbulent simulations. Based on those results, it may be assumed that the observed effect is mainly driven by turbulence.

As the main goal of this work is to find a model representing a hydrokinetic turbine in 2d simulations the results presented above have to be described mathematically. First, Equation 9 and 10 are modified to correctly represent the turbulent results. This is done by replacing the blockage ratio $\epsilon$ with a corrected blockage ratio $\epsilon^*$ with
\[
\epsilon^* = \left(\frac{\epsilon b_{(c_p)}}{1 - \epsilon}\right)^x
\]
(11)

The effect of the aspect ratio at constant blockage ratios on the turbine power appears to be comparatively small (maximum around 3%). But preliminary studies investigating different DAT geometries showed an increase of power of up to 10%. Therefore, the influence of the aspect ratio cannot be neglected. Modelling the influence of this mathematically may be achieved by introducing the exponent $x$ in Equation 11 which is defined as:
\[
x = 1 - \frac{0.45}{1 + (m\alpha)^{-1.6}}
\]
(12)

with $m(c_p) = 0.042$ and $m(a) = 0.083$. Comparing the results from the introduced mathematical model to the solutions of the CFD simulations in Figure 4 it may be noted that good agreement is obtained.

**Figure 4.** Results from adapted turbine model, CFD simulations of AD and DAT

**Figure 5.** Illustration of definition of conversion factor

In order to find out if the developed model is also applicable to DATs an exemplary diffuser geometry is placed in the different channel cross sections already used for the investigations of the actuator disk. The results are also presented in Figure 4. It may be observed that for very small aspect ratios the $c_{p,c}$-curves are nearly superimposed, whereas for lower aspect ratios the curves increasing less steeply compared to the AD results. By adapting the empirical values $b$ and $m$ those results may also be represented by the introduced turbine model.
3.3. Implementation in 2d shallow water solvers

The adapted turbine model is implemented in the in-house shallow water solver tidalFoam described in detail in [5] and [6]. As already mentioned in Chapter 2 the existing turbine model is based on the assumption that the 2d and the 3d turbine thrust is equal. This assumption requires the possibility to model the behavior of the machine through the LMADT, as the determination of the 2d thrust is based on this concept. Considering a diffuser and/or the influence of the channel boundaries this requirement is not fulfilled. As described earlier the introduced models work with a strongly manipulated LMADT and lead to a significant deviation of the determination of thrust. Therefore, the thrust curve of an ideal actuator disk and the one of the actual machine are not superimposed and must not be equated. This is exemplary illustrated in Figure 5. When equating the thrust forces despite the issues stated above, an incorrect cell velocity and consequently a wrong undisturbed infinite velocity $u_\infty$ is determined by the turbine model. In order to correct this, a “conversion factor” $f$ is introduced and defined according to Equation 13.

$$f = \frac{T_{AD}}{T_{DAT,3d}} = \frac{4a_c (1 - a_c)^2 A_d \tau}{c_{p,c} q A_r}$$  \hspace{1cm} (13)

It converts the total 3d machine thrust $T_{DAT,3d}$ to a value on the LMADT thrust curve at same induction factor $a_c$, as illustrated in Figure 5. Unless otherwise defined the conversion is executed at the optimum of the machine. Applying the correction when determining the 2d drag coefficient $c_{d,t}$ for the shallow water solver leads to the following equation:

$$c_{d,t} = \frac{A_d c_{p,c} u_\infty^2}{2A_{cell} \bar{n}^2 (1 - a_c)}$$  \hspace{1cm} (14)

The presented correction of the actual thrust ensures a correct determination of the deceleration caused by the energy extraction through the turbine. The power output of the machine may be calculated according to Table 1 when determining $u_\infty$ from the cell velocity using Equation 2. This may be done in post processing.

Regarding the spatial discretization it has to be noticed that the size of the turbine cell has to roughly represent the dimensions of the modeled machine including the diffuser. For the open channel flow mainly the length in flow direction is important. In case that the turbine cell is e.g. too short the increase of the local water level is overestimated due to the small cell area. Following the conservation of mass this leads to an underestimation of the cell velocity $\bar{n}$ respectively of the undisturbed infinite velocity $u_\infty$.

Furthermore, it has to be noticed that Equation 9 and 10 describe a 1D model, where the influence of the limited cross section is included. In this case a part of the influence of the decrease of the width is represented by the 2d mesh. Therefore, also the 2d thrust should be corrected. But investigations showed that the influence of decreased width in the 2d mesh only plays a role for very narrow channels and even then is comparatively small.

4. Conclusion and outlook

A turbine model considering the influence of channel geometries including the aspect ratio is presented and calibrated through several numerical flow simulations. Also the implementation in 2d shallow water solvers is described. A future step will be to change the block profile, which is - at the moment - used to determine the upstream undisturbed velocity to a 1/7-power law profile and validate the results through 3d flow simulations. Free surface effects as described in [10] and [11] do not have to be include in the theoretical model, as the position of the water surface is calculated and hence considered by the shallow water solver. It would also be interesting to determine the influence of those effects on the power output. Furthermore, it is planned to
investigate different ADT geometries in order to find a way of determining the empirical values required for the adapted turbine model with low effort. Also the interaction of more turbines in park arrangements and the ability of the turbine model to describe this correctly is of major interest.

References
[1] Burton T 2011 Wind energy handbook vol 2. ed. (Wiley) ISBN 978-0-470-69975-1
[2] Ruopp A, Ruprecht A, Riedelbauch S, Arnaud G and Hamad I 2014 IOP Conference Series: Earth and Environmental Science vol 22 (IOP Publishing) p 062002
[3] Shives M and Crawford C 2012 Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 226 112–125
[4] Garrett C and Cummins P 2007 Journal of Fluid Mechanics 588 243–251
[5] Ruopp A 2016 Optimierung von symmetrischen Gezeitenströmungsturbinen und deren Analyse in großräumigen Gezeitenströmungsgebieten Ph.d. thesis Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart
[6] Ruopp A, Daus P, Biskup F and Riedelbauch S 2015 Journal of Renewable and Sustainable Energy 7 061707
[7] Joly A, Pham C T, Andreevsky M, Saviot S and Fillot L 2015 Telemac User Club 2015
[8] Lewis R, Williams J and Abdelghaffar M 1977 Wind Engineering 1 104–125
[9] Nishino T and Willden R H 2012 International Journal of Heat and Fluid Flow 37 123–135
[10] Whelan J, Graham J and Peiro J 2009 Journal of Fluid Mechanics 624 281–291
[11] Pelz P and Metzler M 2013 IOP Conference Series: Materials Science and Engineering vol 52 (IOP Publishing) p 052019