Simulation validation of a tidal turbine: comparison to Southampton’s experiment

Corentin Lothode∗†, Didier Lemosse†, Emmanuel Pagnacco†, Eduardo Souza†, Augustin Hugues∗, Yann Roux∗

∗K-Epsilon, France, corentin@k-epsilon.com
†LOFIMS, France, eduardo.souza@insa-rouen.fr

1 Introduction

In this report, the mean power and thrust are compared between CFD computations and experimental data. The parameters are the flow speed, the blade pitch angle and the rotation speed. Then a surrogate model with a Latin Hypercube Sampling is used to find the optimal pitch angle and rotation speed.

The experiment used is the one made by Bahaj et al. and was the first to be performed on tidal turbines. It contains many results for different pitch and is very useful to compare against. Though the experiment has a high blockage correction (up to ≃ 18%), it is well documented and provide much insight. Many people used this experiment to validate codes, Blade Element Momentum Theory (BEMT) for example in Other experiments exist today but will not be used in this paper (Ifremer, Liverpool, Manchester). BEMT is a good approach to assess the performance of one turbine, but it fails to perform for multiple turbines.

To avoid this problem, other approach has been developed such as the Vortex Lattice Method (VLM) in 7. Their main focus is the wake of the turbine to study the interaction between two or more turbines 8. Their results are good until stall which is expected since their method force the flow to be attached until the trailing edge. Later included turbulence.

Attempts to use Computational Fluid Dynamics (CFD) on wind or tidal has been performed in the past. To avoid too much computational efforts, many authors modeled the behavior of the turbine instead of resolve the full geometry. For instance, has used Large Eddy Simulation (LES) with the turbine replaced by an approximated model of a concentrated drag force to study the wake development. Also using an approximated model for the turbine, performed a LES computation using an actuator disk.

Fully resolved blade geometry CFD computations are computationally expensive, but can give many more insight about the flow behavior and force distribution along the blade. compared \( k – \omega \) SST, Launder-Reece-Rodi turbulence model (LLR) and LES on the 20° pitch angle case of as an unsteady simulation, including the mast and a simplified geometry of the cavitation tunnel. The results are very interesting but lack other pitch angles to further validate the models which is what we are addressing in this paper.

<table>
<thead>
<tr>
<th>Name</th>
<th>Property</th>
<th>Name</th>
<th>Property</th>
</tr>
</thead>
<tbody>
<tr>
<td>( u )</td>
<td>velocity</td>
<td>( p )</td>
<td>pressure</td>
</tr>
<tr>
<td>( V )</td>
<td>inlet velocity (m/s)</td>
<td>( \rho )</td>
<td>density of the fluid (kg/m³)</td>
</tr>
<tr>
<td>( r )</td>
<td>radius of the turbine (m)</td>
<td>( S = \pi r^2 )</td>
<td>tidal turbine area (m²)</td>
</tr>
<tr>
<td>( Q )</td>
<td>turbine torque (Nm)</td>
<td>( T )</td>
<td>turbine drag (N)</td>
</tr>
<tr>
<td>( \Omega )</td>
<td>turbine rotation speed (rad/s)</td>
<td>( TS R = \frac{\Omega r}{V} )</td>
<td>tip speed ratio</td>
</tr>
<tr>
<td>( C_p = \frac{P}{\frac{1}{2} \rho V^2} )</td>
<td>pressure coefficient</td>
<td>( C_p = \frac{Q \times TS R}{\frac{1}{2} \rho V^2 r S} )</td>
<td>power coefficient</td>
</tr>
<tr>
<td>( C_t = \frac{T}{\frac{1}{2} \rho V^2 S} )</td>
<td>thrust coefficient</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 1: Notations used

2 Description of the experiment

The experiment is fully described in The tests were carried out in a cavitation tunnel at Southampton Institute (see Figure).

The rotor diameter of the turbine is 800mm. It was chosen as a compromise between maximising Reynolds number and not inducing too much tunnel blockage correction. The blockage correction is based on an actuator disk model of the flow through the turbine in which the flow is presumed to be uniform across any cross section.
of the stream tube enclosing the turbine disc [13]. For example, with a single rotor and a thrust coefficient of 0.8, the corrections amounted up to 18% decrease in power coefficient and 11% decrease in thrust coefficient for the cavitation tunnel and up to 8% and 5% decrease, respectively, for the towing tank.

The blades are made out of the NACA 63-8xx serie. The distribution of pitch and thickness can be found in [1]. We kept the values used in [1] meaning that the pitch distribution is in fact the pitch of the element at radius 80mm (15° means taking the blade as the original blade pitch, 20° means imposing 5° pitch to the blade). Many tests were performed : varying the tip immersion, the blade pitch angle and yaw angle. In this study, mainly 5 batch are of interest for this study and are reported in Table 2.

3 Description of the simulation

The solver is FINE/Marine™which is distributed by NUMECA International. It is developed by the LHEEA laboratory. It solves the Reynolds-Averaged Navier-Stokes Equations in a strongly conservative way. It is based on the finite volume method and can work on structured or unstructured meshes with arbitrary polyhedrons [14]. The velocity field is obtained from the momentum conservation equations and the pressure field is obtained according to the incompressibility constraint. The pressure-velocity coupling is obtained using the SIMPLE algorithm. All the variables are stored in a cell-centered manner. Volume and surface integrals are evaluated according to second order schemes. The time integration is an explicit scheme of order two. At each time step, an internal loop is performed (called a non-linear iteration) associated with a Picard linearization in order to solve the non-linearities of the Navier-Stokes equations.

The equations are formulated according to the Arbitrary Lagrangian Eulerian paradigm and therefore can easily take into account mesh movements. In order to be able to rotate the geometry, we are using the sliding interface capability [15] of ISIS-CFD (see Figure 2b). Several turbulence models are implemented in ISIS-CFD. The turbulence models are here to avoid resolving completely the Navier-Stokes equations. In the case of LES, a low-pass filter is used to avoid resolving the smallest scale in space and time. For RANSE an additional viscosity term called eddy viscosity $\nu$ allows to use even less discretise grids and bigger time steps. In this study, we used the RANSE using the SST-$k-\omega$ model [16].

The computational domain is made of a box of width $L_x = 4m$, length $L_y = 6m$ and height $L_z = 4m$ (see Figure 2a). The size of the cylinder subdomain which is rotating is $L'_y = 0.6m$ and $r' = 0.6m$. The sides and outlet boundary condition use a zero-gradient boundary condition for both velocity and pressure. The inlet boundary condition is an imposed velocity of $u_{\infty}$. The rotation velocity is added to the cylinder through the ALE formulation. The mesh has 4.1 million cells. Refinement boxes were placed to capture the tip vortices and the wake correctly.

4 Results discussion

For a blade pitch angle of 15°, the blade sections show a higher angle of attack compared to the blade pitch angle of 20°. As stated previously, the blade pitch angle of 20° is the design angle of the turbine. In other words, it mean that the blades apparent angle of attack is higher than for the design pitch angle of 20° by 5°. The apparent angle of attack is reduced by having an higher rotation. Hence to have the expected flow behaviour, the turbine will have to rotate faster than the design rotation speed in order to compensate the blade pitch angle. The pressure plots are showing the expected behaviour. The flow around the blade is fully separated at a TSR of 3 and still partially detached for TSR from 4 to 5. For a TSR ranged from 4.5 to 5.5, the error observed between the experimental data and the simulation is about 5% for the $C_p$ and 2% for the $C_t$. Outside this range, the error is bigger, especially for high TSR where the $C_p$ drops a lot faster than for the computation. The power
coefficient by section shows that all the blade is propulsive for TSR from 4 to 6. For TSR 7 and 8, the tip starts to generate more drag than it generates propulsion. The observed $C_p$ peak is observed for a TSR of 5, which is the result for the experiment, only the value differs (0.4626 for the simulation, 0.44 for the experiment).

For the design pitch angle of 20°, the flow is only partially detached for a TSR of 3 on the upper part of the blade, and fully separated for the other half of the blade. Some detached flow can still be seen for TSR up to 4.5, but is limited to a small area of the blade. The error made between the experimental and the simulation results is very small (about 1% for both $C_p$ and $C_t$). It is probably due to an accurate blocage correction, and a nice flow behavior around the blades. The power coefficient by section shows that the whole blade is propulsive for all tested TSR, showing that the design is correct. The $C_p$ peak is not as clear as for an angle of 15°, and occurs at a TSR of 5.5 ($C_p = 0.4533$), although the value obtained for a TSR of 5 and 6 are really close (0.4450 and 0.4531 respectively).

Blade pitch angles of 25° is starting to show a significant loss in term of power spike, and it is even worse for blade pitch angles of 27° and 30°. For the angle of 25°, the flow behaviour is similar to the blade pitch angle of 20° with only the lower third part of the blade showing separation. For blade pitch angles of 27° and 30°, this separation is even lower. For blade pitch angle of 30°, the flow starts to separate at the tip, on the front side. The difference observed between the simulation and the experiment is only significant for the blade pitch angle of 30°, otherwise the agreement is good (less than 5% difference). For blade pitch angles of 25°, 27° and 30°, the $C_p$ peaks at a TSR of 5, 4 and 4 respectively, with a $C_p$ of 0.3491, 0.3012 and 0.2391. The power coefficient by section has progressive behaviour, the higher the blade pitch angle is, the faster the tip part of the blade stops generating power and starts to be counterproductive.
5 Optimization of two parameters (TSR and pitch)

The goal is to optimize the power coefficient with two parameters (TSR and pitch). The first thing is to generate the couple of parameters in order to use a maximum of space. For this, a Latin Hypercube Sampling algorithm is used, this algorithm is base on the principle of the latin square, then it is extended to more dimensions. (The principle of the latin square, is that you plan is divided in square, and there one and only one value in each rows and each columns). A good sampling of the space is important, because we will see after that the evolutionary algorithm will have good results only on the area define by the sampling points, beside this area, the results are false. Scripts automatically mesh and setup the simulation of all the set of parameters obtained during the sampling.

![Couple of values obtain with LHS algorithm](image1)

![Response Surface from Dakota](image2)

We then obtain a power coefficient for each couple (TSR, pitch), and from this we use a response surface method based on a Gaussian process [17]. The response surface allows to use an optimisation algorithm on it without having to re-run computationally expensive simulation. Here, the optimization of the power coefficient values are calculated with an evolutionary algorithm.

The figure [4b] shows the surface obtained for the turbine. We have now a continuous surface, with only 20 couples of parameters at the beginning. The best parameters obtained through this process are 19.5° for the pitch and a TSR of 5.85. The calculation with those two optimum parameters shows an error of 7% between the calculation and the value on the surface. From this, an iterative process can be run by adding the new point to the set of parameters to obtain a new response surface and compute a new set of optimum parameters. After a few iterations, the optimum set of parameters should be found.

Conclusion

CFD results of a tidal turbine are shown for various conditions and are compared to experimental data. The results of the simulations show a very good agreement with the experiments. From these preliminary results, an optimisation methodology is developed with a surface response method coupled to a genetic algorithm. These initial results are promising and a follow up study, optimizing the reliability of the turbine subject to random current speeds and directions or the dynamic response of the blades with fluid-structure interaction would be interesting to pursue.

References


