EFFECT OF THE VELOCITY PROFILE OF INCOMING FLOW ON THE PERFORMANCE OF A HORIZONTAL AXIS TIDAL STREAM TURBINE

Cora Fung[1,2], Stuart Walker[1,2], Robert Howell[2]
[1]E-Futures Doctoral Training Centre, University of Sheffield
[2]Department of Mechanical Engineering, University of Sheffield

SUMMARY: Compared to solar and wind power generating technologies, tidal stream technology is relatively new and there are many problems that hinder the deployment of tidal stream turbines. One such problem is the effect of the tidal stream velocity profile resulting in varying velocity magnitude and its effect on turbine performance. A study incorporating experimental data and computational fluid dynamics (CFD) simulation is carried out in order to start to understand such effects. A CFD model based on an existing turbine was created and validated by comparing its results against experimental results of a previous study. A velocity profile was measured in a water flume experiment, which was then applied to the CFD model as a new upstream boundary condition. The results show that, due to the variation in flow velocity, the torque generated by each blade will fluctuate as it travels through a complete rotation. Despite this, the trend of torque distribution on individual blade has very little change throughout the rotation. Similar effect also takes place on thrust loading of the blades, meaning that blade deflection will change as the turbine rotates, inevitably affecting the fluid dynamics of the turbine.

Keywords: Tidal stream turbine, tidal energy, CFD, turbine performance, velocity profile

INTRODUCTION

Renewable energy has become a popular research topic in recent years due to concerns over climate change and energy security. Technologies such as wind turbines and photovoltaic panels have since matured and are now widely implemented. Tidal power technology is still in its infancy and is far from reaching its full potential. Taking the U.K. as an example, according to government statistics, 47% of electricity generated from renewable sources was delivered by wind turbines in 2012. Although solar PV systems only accounted for 3% of total renewable electricity generation, it had increased by 387% since 2011. As the UK is surrounded by the ocean, it is in abundance of tidal energy; yet, shoreline wave and tidal power only provided 0.01% of electricity combined [1] .

Tidal power has major advantages compared to wind and solar power due to its predictability – whilst the amount of available solar and wind energy depend heavily on the weather, the time and speed of incoming tides at particular locations are more certain. The power generation capacity per unit area occupied of tidal stream turbines is high when compared to that of solar, wind, and other conventional tidal power technologies (e.g. barrage)[2]. There is also the flexibility of forming tide farms of different sizes, at different locations where times of peak flow (and so power generation) offset each other, forming a more stable power source which is highly suitable to be used as base load.

Despite the advantages of tidal turbines, there are various problems and unknowns that hinder the development of the technology, one of which is the effect of the velocity profile of incoming flow with varying velocity magnitude on turbine performance. As it is not desirable to deploy tidal turbines in complete free stream flow due to physical constrains, studying such effects will help to understand how turbines operate in real tidal flow. To achieve this, a study incorporating CFD simulation and experimentation was conducted. A CFD model of an existing turbine was first created and validated. The velocity profile was measured through a water flume experiment, which was then used in the CFD study.

PRELIMINARY WORK

Validation of the CFD model

The turbine used in these simulations was previously studied by Bahaj et al[3,5]. The turbine is 800 mm in diameter, with varying pitch angle and thickness along the span of the blade. All boundary conditions used in the CFD simulation are the equivalent of the experimental setup published in [5].

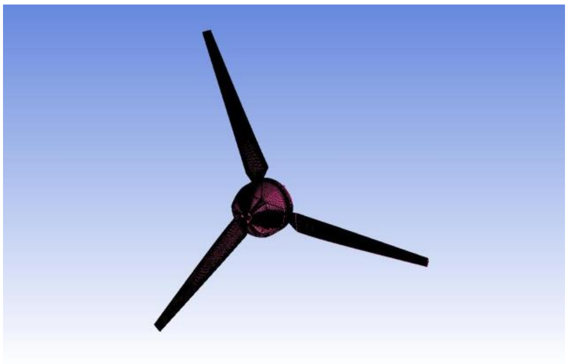

Figure 1 Mesh of the turbine in the CFD simulation

Before constructing the final model, a series of studies were conducted on the 2D blade section and the simplified 3D model in order to ensure the accuracy and efficiency of the simulation [4]. All geometries and computational domains were constructed in the meshing software ICEM, in which a multi-block meshing strategy was deployed to create structured grids.

To validate the accuracy of the CFD model, the power coefficients (c_p) of a range of tip-speed ratio (λ) were recorded and compared against experimental results, which are shown in figure 2. The CFD model had predicted an earlier and more rapid stall, which took place at TSR 5. Despite the differences, the two sets of data lie within the same range and the overall trends have similar characteristics. The accuracy of the CFD model should be sufficient for the purpose of this study.

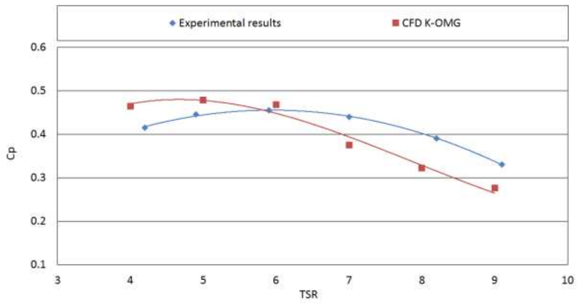

Figure 2 Experimental and predicted power coefficient vs. tip-speed ratio

Measuring upstream velocity profile

As part of an experimental project being undertaken using the large tilting water flume at the University of Sheffield Sediment Transport Laboratory, a velocity profile was recorded for use during this work. The profile was recorded using Laser Doppler Velocimetry (LDV) equipment, in a water depth of 300mm. The water flume has a width of 500mm, and the measurement location was 9m downstream of the flume inlet, at which point a fully turbulent boundary layer was confirmed to exist. A photograph of the flume and the LDV measurement equipment in use is shown below.

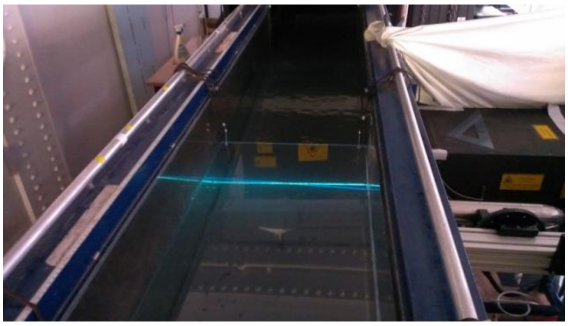

Figure 3 A photograph of the water flume facility

A flow profile was recorded at a bulk flow rate of 40l/s, giving a Reynolds number $Re=3x10^5$ and Froude number Fr=0.062.

The profile was recorded by taking flow velocity measurements at 21 positions along an imagined vertical line from the flume base to the water surface, at the horizontal centerline, 9m downstream of the inlet. Velocity measurements were taken at 14mm intervals from 20mm above the flume base to the water surface. At each position, LDV equipment was used to record 3000 data points at a frequency of 90Hz, giving a measurement of velocity over around 33 seconds at each position.

A two channel LDV system was used in this study, made up of a TSI control unit and Photomultiplier Tube (PMT), Melles Griot 43 Series Argon Ion Laser, TSI Aerometrics beam splitter and TSI remote laser head unit. LDV seeding particles were used to ensure reflection of the laser beams. The seeding particles used were silver-coated glass spheres of 17μm diameter (brand name "Conductospheres M-18", manufactured by Microsphere Technology). A volume of 50cm³ was used, as recommended by the LDV equipment manufacturer, giving a seeding density of 3 parts per million (ppm).

The LDV system was operated at a PMT voltage of 800 V and burst threshold value of 100 mV. PMT voltage controls the gain applied to the signal from the photodetector, with a higher voltage giving greater amplification of individual particle response signals. However, since this gain is applied to the signal as a whole, care must be taken to avoid using excessively high values, as this can result in signal distortion. The burst threshold is the value of signal strength at which a signal is recognised by the PMT unit as a viable response, rather than noise. These values were selected iteratively in order to maximise the performance of the data recording, which was monitored using data rate and burst efficiency statistics. Data rate describes the number of signals received by the PMT each second which are above burst threshold strength, and burst efficiency is a measure of the number of these signals to the number of emitted triggers. Guidance for monitoring acceptable levels of both measurements was sought from equipment manufacturers. Finally, a band pass filter was set between 0.03 MHz and 0.3 MHz in order to avoid any erroneous signals outside this range. The range was validated by plotting a frequency histogram during the analysis. The clearly visible bell-shaped curve which was observed confirms that this filter range does not remove any valid data. Data rates of between 180 and 300 Hz and burst efficiencies of around 67% were observed during the velocity profile measurement.

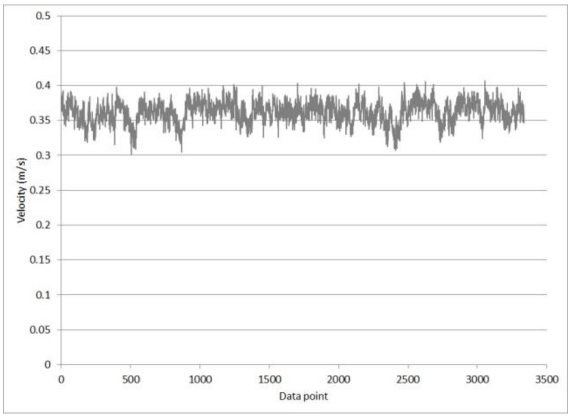

Figure 4 Sample LDV velocity-time series

Variation in velocity with time at each recording location was low. A sample of velocity recorded over a 33-second measurement period is illustrated above. This velocity series was recorded at 244mm above the flume base.

SIMULATION SETUP

A curve was fitted to the data obtained from the water flume in order to create the velocity profile which was subsequently applied to the inlet boundary of the computational domain. The profile and the dimensions of the computational domain were scaled according to the size of the turbine in the CFD simulation and the water flume experiment setup.

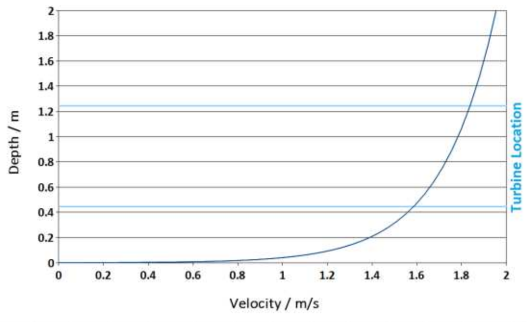

Figure 5 Velocity profile at inlet boundary

The computational domain was split into two parts: a stationary domain and a rotational domain. The former represents the free stream of incoming flow, whilst the latter represents the rotating turbine, which revolves around the x-axis at every time step.

To simplify the simulation, the top, bottom, and the two sides of the domain were modeled as walls. The downstream boundary of the domain was defined as pressure outlet, which allows recirculation to be captured. Downstream length was extended, which allows future study on wake recovery.

For this study, the model was run at a λ of 4. To ensure stability of the simulation, an implicit calculation method was used with a time step which is equivalent to 0.125° of rotation. In order to investigate the changes in torque, torque distribution of the turbine was recorded throughout the rotation.

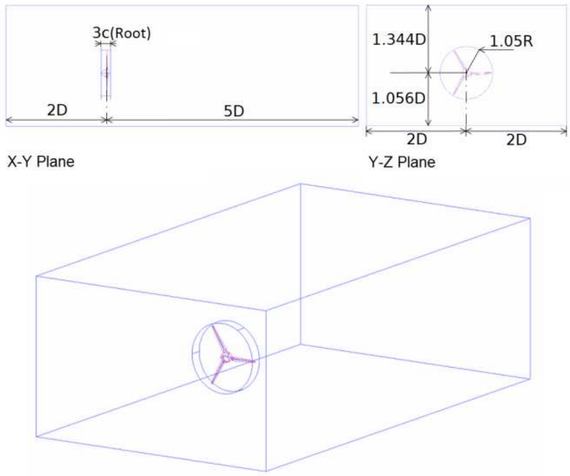

Figure 6 Dimensions of the computational domain

RESULTS AND DISCUSSIONS

Figure 7 shows the torque generated along the span of the turbine blade at different location of the cycle. 0° indicates a blade pointing directly upwards, whilst 180° refers to the position where the blade pointing directly towards the bottom. The results show a fluctuation in torque generation as the turbine rotates. The blades give higher torque when approaching the top of the cycle and lowest torque at the bottom, which is where the flow velocities are at a maximum and minimum respectively.

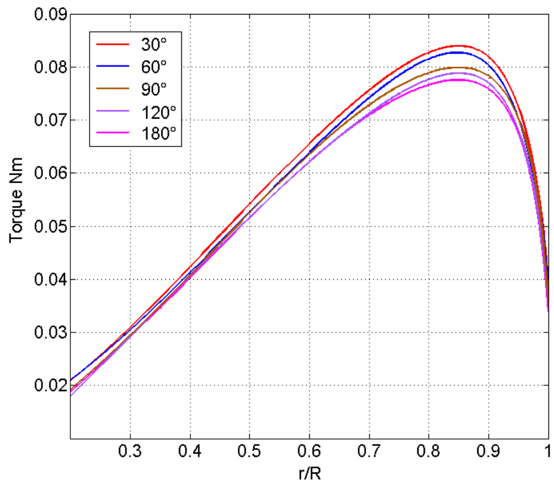

Figure 7 Torque generation vs. blade length

It was expected that torque generation would increase towards the tip of the blade due to higher relative velocity; however, the peak torque generation takes place at 0.86R of the turbine, and that torque generation dropped rapidly at blade tip. This is explained by the downwash effect at the tip, which causes a drop in lift generation. Despite the variation in free stream velocity, the general trend of torque distribution on the blade has not changed significantly. The location of peak torque generation is constant throughout the rotation, although the torque around this area has the highest fluctuation.

The contour plot in figure 8 shows the torque distribution on the suction side of the turbine. It can be seen that the general trend of torque distribution on the blades are highly similar throughout the rotation. However, on closer inspection, the blade which is nearest to the bottom has less negative torque towards the root. This means that as the blade travels towards the bottom, the pressure gradient of the blade section at the root becomes less adverse.

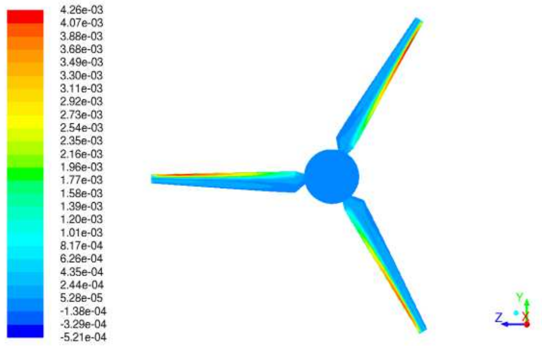

Figure 8 Pressure torque distribution on the suction side of the turbine

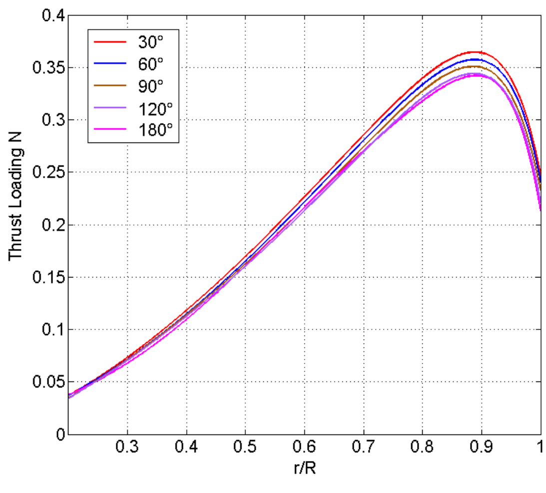

Figure 9 Thrust loading on the blade vs. blade length

Figure 9 shows the thrust loading acting on the turbine in the axial direction. Thrust loading will cause bending and displacement of the turbine blades. This blade displacement will affect the fluid dynamics of the turbine, which in turn will affect thrust loading and blade displacement. It should be noted that this coupling of the fluid dynamics and structural changes can only be studied with a fluid-structure interaction model, and is not captured in the current model. Therefore, the thrust loading predicted by this CFD model only gives an indication of the level of thrust loading to be expected at this flow condition.

Once again, due to the variation in flow velocity, this axial thrust force is larger at the top of the cycle and smaller at the bottom. Peak loading on the blade takes place at 0.89R of the turbine. The displacement of the blade will vary according to the properties of the material used to construct the blades, but could be estimated using finite element analysis.

CONCLUSIONS AND FUTURE WORK

This paper introduced a study that investigates the effect of non-uniform incoming flow on the torque generation of a tidal stream turbine. Both the experimental and computational aspects of the study were explained in detail.

Results of the study were presented and discussed. Torque generation and thrust loading of each blade varies throughout the rotation cycle due to the uneven free stream velocity. However, the changes in torque and thrust loading distribution along the blades only had minor changes throughout the cycle.

For future study, simulations of the same setup will be conducted at different TSR in order to investigate performance of the turbine. Information on wake recovery could also be gathered from this simulation setup.

REFERENCES

[1] I. MacLeay et al., *Digest of United Kingdom Energy Statistics 2013*, Department of Energy and Climate Change, National Statistics: London

[2] D. MacKay, “Tide”, *Sustainable Energy - Without the Hot Air,* 2008, UIT Cambridge Ltd.

[3] A.S. Bahaj et al., “Power and thrust measurements of marine current turbines under various hydrodynamic flow conditions in a cavitation tunnel and a towing tank”, *Renewable Energy*, **32**, 2007, pp. 407-426.

[4] C. Fung et al., "Development of a Computational Fluid Dynamics Simulation Model for a Horizontal Axis Tidal Stream Turbine", Proceedings of University of Sheffield Engineering Symposium, 2014 (in press)

[5] A.S. Bahaj et al., “Experimental verifications of numberical predictions for the hydrodynamic performance of horizontal axis marine current turbines”, *Renewable Energy*, **32**, 2007, p.p. 2479-2490.