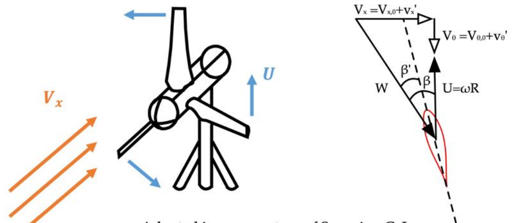
Alleviating Mechanism for Unsteady Load in Tidal Turbines

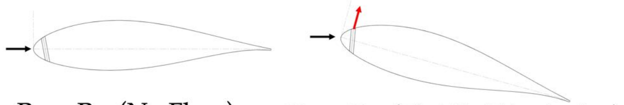
Stephen J. Sun, Dr. J. R. Farman, Dr. J. V. Taylor
Email: js2597@cam.ac.uk

Problem Statement



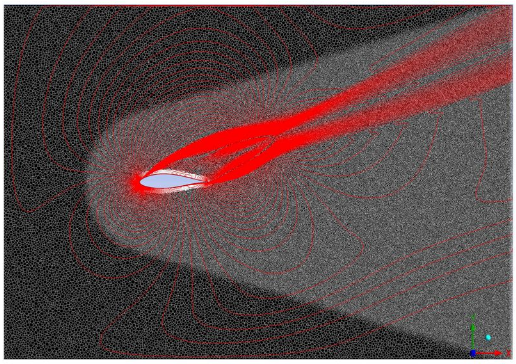
Adapted image courtesy of Sequeira, C. L.

Tidal turbines experience oceanic gusts. Fluctuations in velocity translate to fluctuating flow angle, which changes the loads on blades, generator and pylon. A flow passage connecting blade surfaces is used to adjust the flow around the blade, aiming to reduce the effect of the fluctuations in blade forces.



$P_{SS} \approx P_{PS}$ (No Flow) $P_{SS} < P_{PS}$ (Fluidic Trip Active)

CFD Simulation



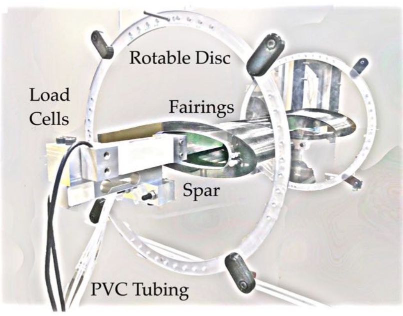
High order computational method uses a custom numerical grid for the finite volume approach, to accurately capture the wake behind the blade, the pressure distribution as well as the mass flow rate through blade passage. This is to quantify its effectiveness when various design parameters are changed.

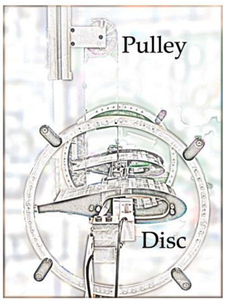
Experimental Design

Test blades mounted onto a rotatable spar in a customised tunnel exit section.

Key metrics that are monitored:

1. Blade forces (using load cells)
2. Surface pressure around blade
3. Ambient temperature and pressure (which also provides the density)

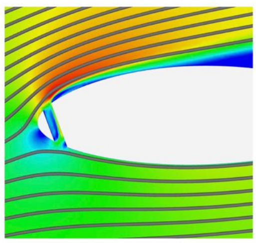


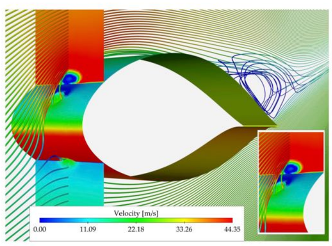


Results Summary

Improved designs validated with CFD and experiment. Parametric studies indicate effects including the following:

1. Suction surface opening location can be moved for either boundary layer injection or suction peak reduction;
2. More span-wise spacing introduces non-uniformity and surface vorticity;
3. Mass flow inside thin passages can be approximated with viscous pipe flow.





UNIVERSITY OF CAMBRIDGE
Whittle Laboratory

Tidal Turbine Load Alleviation – Case Study Report

Stephen Sun

Horizontal axis tidal-stream turbines are becoming increasingly prevalent. However, marine flow unsteadiness poses a challenge to structural reliability of blades and drivetrain. The issue with fatigue loading is exacerbated compared to horizontal axis wind turbines despite similar aerodynamic design principles. This is due to the higher thrust and torque levels on the blades.

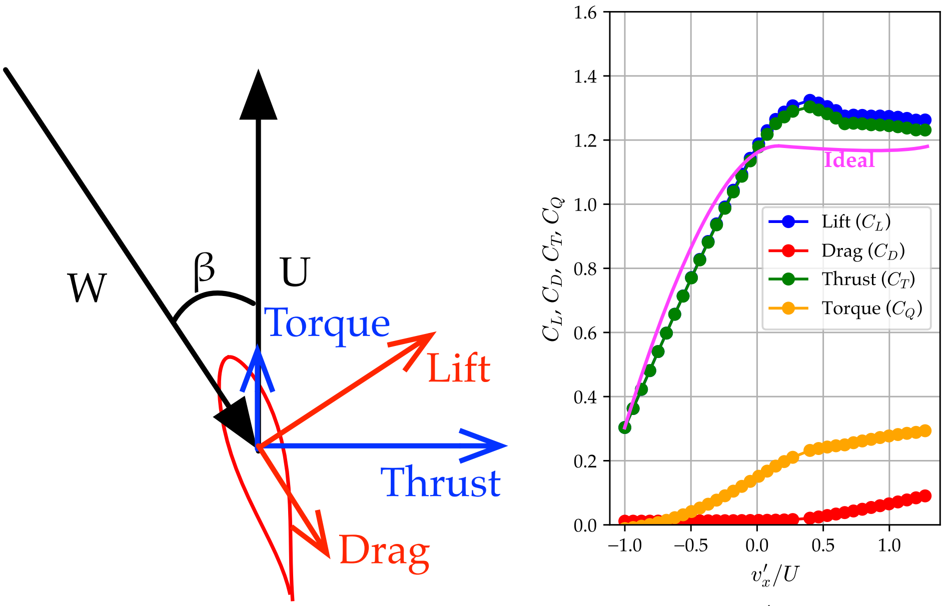


Figure 1: Key loading parameters of a tidal turbine blade (span-wise view)

Most of the unsteadiness arises from fluctuations in stream-wise flow direction. For this reason, the majority of oceanic gusts can effectively present to blades as fluctuations in incidence. Previous research at University of Cambridge shows the dynamics of load fluctuations can be approximated by how the lift changes with incidence at steady state with low errors. By reducing the sensitivity of lift towards varying incident flow angles around design point of the blade, the unsteady loading can be alleviated, thus reducing fatigue on the device overall.

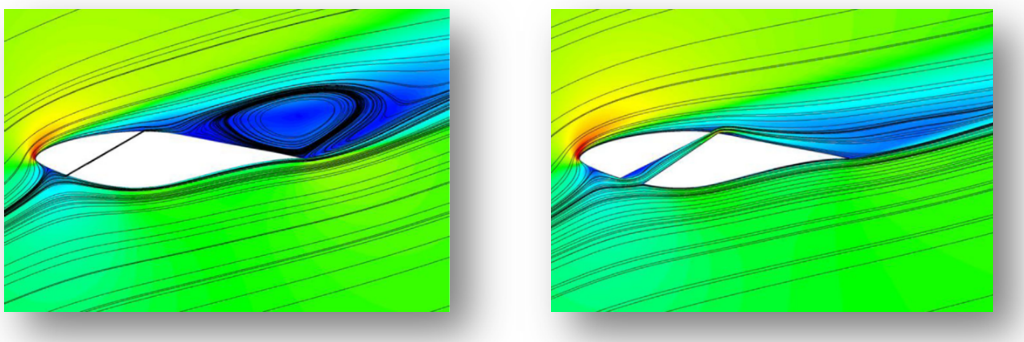


Figure 2: Slot-based passive flow control, images courtesy of Belamadi, R.

To achieve the lift moderation, past studies from external institutions explored use of various flow control devices. Specifically, active flow control involves using a sensor-actuator loop to adjust the blade geometry. However, complexity and strong dependence on how accurate the sensor can be in measuring the flow field can make these devices less desirable. This case study proposes the use of a novel passively controlled mechanism with blowing and suction jets through a common passage within the to get the desired level of alleviation.

Compared to a conventional slot-based approach, this has the advantage of being more versatile by not sacrificing blade structural integrity as much, and introducing span-wise non-uniformity (and potentially vorticity) to allow for more design space. This study broadly explored this expanded design space.

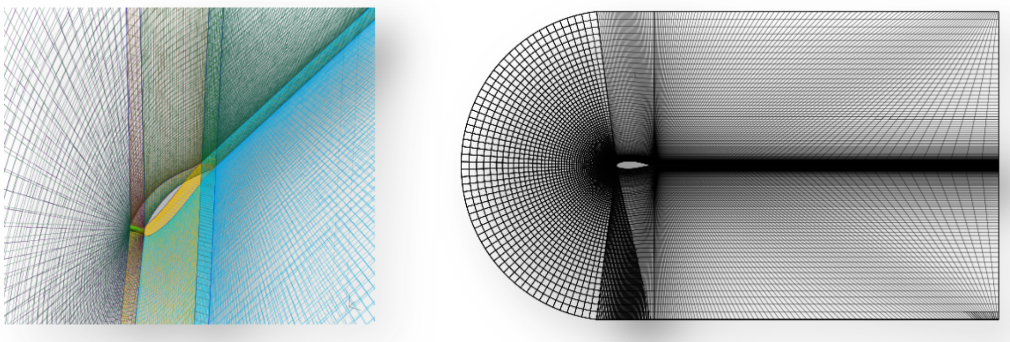


Figure 3: Initial grids used for numerical CFD simulations

The early lower order simulation model used XFOIL to characterise the surface pressure distribution of the datum blade. This allows for a preliminary understanding of the likely chord-wise locations of passage inlet and outlet on the pressure and suction sides. A higher order computational fluid dynamics (CFD) model is then developed to provide a more detailed understanding of the flow field within the passage and surrounding the blade section. This process is optimised by a series of mesh iterations to improve accuracy. Specifically, this includes an ability to resolve flow within the viscous sublayer, as well as inside the thin passages. Once the suitable model was identified, Pythonic automation was used to conduct a series of parametric studies exploring the design space. The 3 key variables investigated include the location of suction surface opening, the span-wise separation of the passages, and their cross-sectional size.

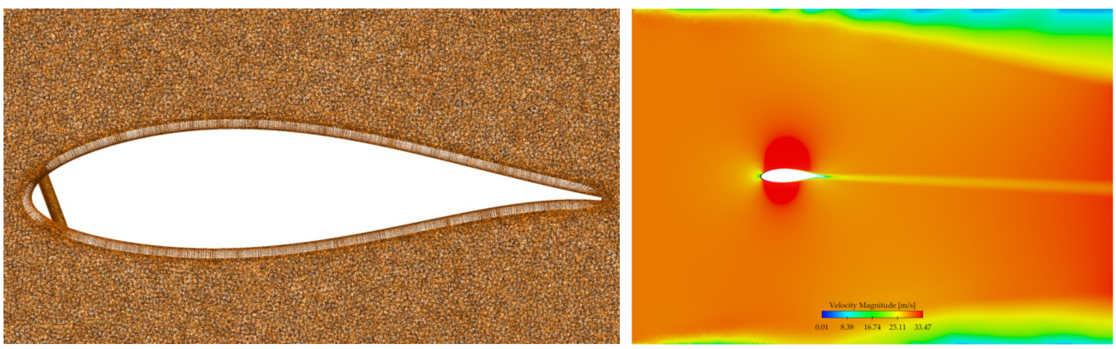


Figure 4: Zoomed in view around the blade; and open-jet simulation results

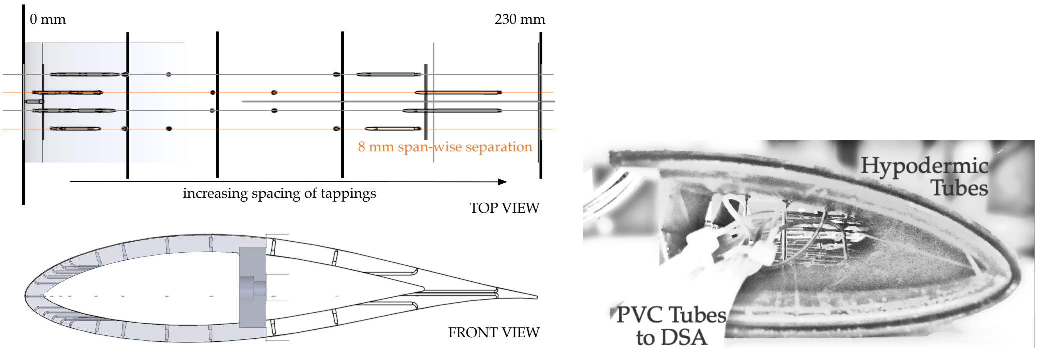


Figure 5: Design of pressure tappings and actual assembled blade with hypos

To validate the CFD, experimental tests were conducted following the same parametric setup. It was found that the flow within the passage can be accurately predicted by theory of viscous pipe flow when the passage diameter is small compared to chord length. The mass flow rate is primarily driven by the pressure differential at the openings on both surfaces. A 3D serrated trip like behaviour can also be recovered by ensuring sufficient span-wise separation between the passages. In general, it was observed that larger passage sizes and reduced span-wise spacing can improve the effectiveness of the mechanism, indicating the potential for future throttle-like integration. However, changing positioning of the suction surface opening can drastically alter the mechanism. By placing the opening further aft, the fluidic trip behaviour is replaced by a more conventional boundary layer injection. Above approximately 40% chord, this can lead to an increase rather than decrease in lift, but is likely to retain the benefit of delaying and controlling the onset of dynamic stall.

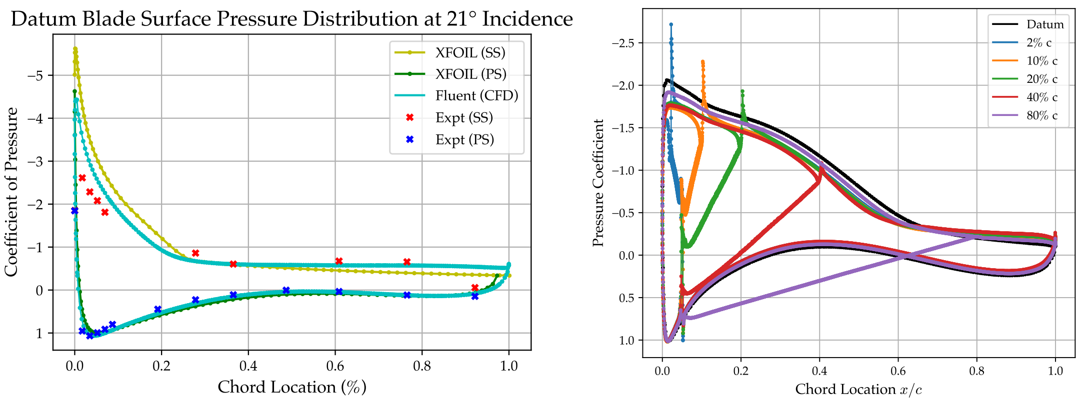


Figure 6: Sample results from Xfoil and CFD on surface pressure distribution

This research has a strong potential to be used in the industry in the future. More effective flow control mechanisms are being developed, and these advancements in technology would reduce the investment and operating costs of the tidal and wind turbines, bringing us one step closer to achieve net zero.

The results also contained 180 GB of simulation data, including mesh, case and solution files. They can be made available to prospective students upon request.