Development of a Hydrokinetic Turbine for Puerto Rico

Andrea Rivera Rodríguez, Estefan L. Diaz Mercado, Prof. Jose R. Pertierra
Polytechnic University of Puerto Rico – Undergraduate Research Program
2019-2020

TITLE V STEM
POLYTECHNIC UNIVERSITY OF PUERTO RICO • SAN JUAN
BRIDGES TO STEM SUCCESS
P 0 0 3 1 C 1 6 0 1 4 1

Abstract

The island of Puerto Rico has struggled for decades with antiquated power plants that produce excess greenhouse gases. In 2017, the delicacy of the overall power system on the island was demonstrated when two Category 5 hurricanes left most of its citizens with no electricity for months. For this investigation, the researchers developed, designed, and simulated a hydrokinetic turbine utilizing real life coastal parameters of Puerto Rico to find out if it's possible to generate electricity with this technology. Per the NOAA database Caricoos, the highest average water velocities of Puerto Rico are found near Vieques, with only 0.16 m/s. For this reason, it was decided early in the investigation to incorporate a duct to maximize flow. With successful European turbines used as a basis, a concept of what would be suitable for the island was developed. Using XFLR5 for airfoil analysis, the Eppler 385 was selected based on its favorable lift and drag data. Utilizing the BEM and BEMT theories, a duct developed specifically for tidal turbines, and initial blade angles for twist, a MATLAB program was created to optimize the 7.4-meter turbine and produce performance results. It output 91.2 Watts of power and an efficiency of 103.7% running at an optimum Tip Speed Ratio of 5 and a RPM of 2.065. Employing Ansys Fluent to validate the results, the turbine produced 77 Watts of power while running with an efficiency of 87.6% at the same TSR and RPM, exceeding the Betz limit of 60%. Both results greatly surpassed the MATLAB run with an unducted turbine that resulted with 47.1% efficiency, proving the profound effect of a duct on turbine performance. Despite the low power output, the researchers managed to produce power with the low current velocities of Puerto Rico, pointing to a possible renewable future via hydrokinetic turbine applications.

Introduction

Before embarking on an investigation of this type, many factors had to be researched prior to committing fully to the design and simulation of a certain type of turbine. It was decided to focus mainly on European projects and companies that have gone through extensive testing or have been functioning well for an extended period. The implementation of any marine energy technology depends on the behavior of the body of water in which it is placed. Using the CARICOOS data base as reference, it was found that the Vieques/Fajardo buoy on the Northeast coast of Puerto Rico has the highest average velocity with 0.16 m/s and that on very rare occasions it has reached 0.50 m/s in average velocity. This velocity is lacking in comparison to the average velocities (~2 m/s) of areas where turbines have been implemented.

A major reason for the limited tidal energy implementations is generally due to low velocities. This brings up the following question: Can there be a turbine that generates power while functioning in a very low velocity area?

Methodology

1. Research implemented marine energy projects.
2. Investigate the coastal conditions of Puerto Rico.
3. Develop a concept with preliminary dimensions and the inclusion of a duct.
4. Utilize XFLR5 to find a suitable airfoil with favorable Cl/Cd over various Reynolds numbers.
5. Create a MATLAB program that optimizes the concept using a combination of duct and blade analysis modified with Blade Element Momentum Theory (BEMT), the selected airfoil and PR water conditions. The following were utilized as inputs for the program
   - Water velocity of 0.16 m/s.
   - Diffuser Area Exit / Area Inlet = 1.89
   - Eppler 385 Data from XFLR5
   - Diffuser exit angles
6. Use the MATLAB results to create a 3D CAD of the turbine and duct utilizing SolidWorks.
7. Run CFD simulations with Ansys Fluent to validate MATLAB results.

XFLR5

After creating a concept, the first step is to select an airfoil. Various airfoils from hydrokinetic turbines, marine applications and airfoil databases like UIUC Database were selected for testing with the XFLR5 software. With a length of 3.7 meters and Reynolds number ranging from 0 – 1,000,000, the lift, drag and initial angle of attack were determined. The Eppler 385 was selected because produced the highest lift to drag ratio with respect to angle of attack as seen on the graph.

Figure 1: Cl/Cd Graph from XFLR5

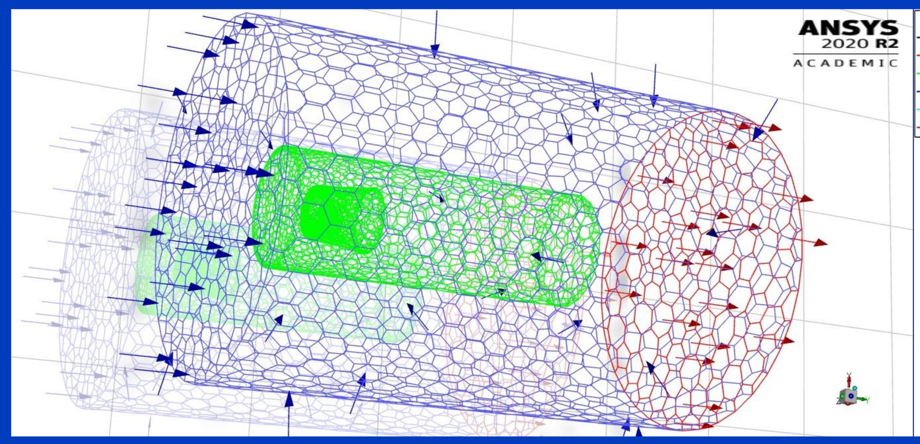


Figure 2: Polyhedral Mesh in Ansys Meshing

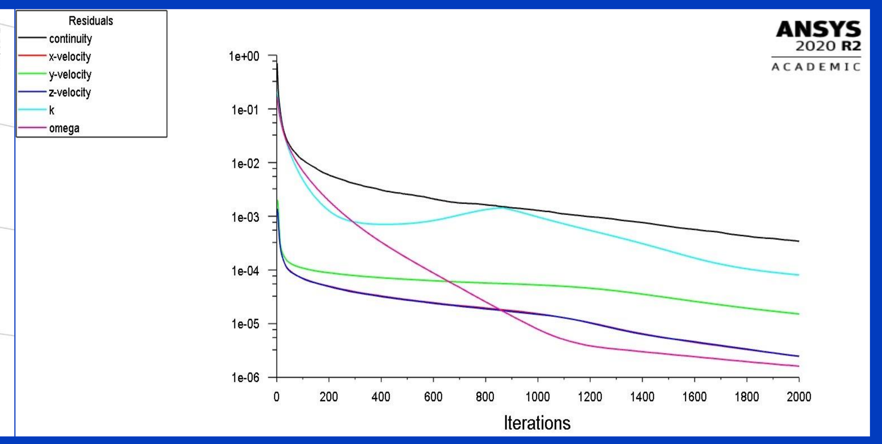


Figure 3: Residuals Graph in Ansys Fluent

MATLAB & SOLIDWORKS

With the airfoil selected, the next step required an advanced MATLAB program that produced both the optimum blade angles for twist per station utilized for SolidWorks and performance results, which will be covered later. This is combined with optimized duct dimensions to create the three bladed, 7.4-meter turbine with the Eppler 385 airfoil and a flat hub. The hub was designed with small dimensions to reduce drag on the CFD.

ANSYS FLUENT

To run CFD simulations of the turbine, Ansys Fluent was selected for its compatibility with turbines. The Meshing process is by far the most difficult part, requiring multiple domains and tedious selection of the mixing planes to ensure the flow is behaving naturally as it passes through the duct and turbine. As seen in Figure 2, the domain for the turbine must be large for the water flow to be at a steady state (0.16 m/s) both before and after exiting the turbine. Taking these steps during the setup of the CFD guarantees a smooth residual value graph like in Figure 3.

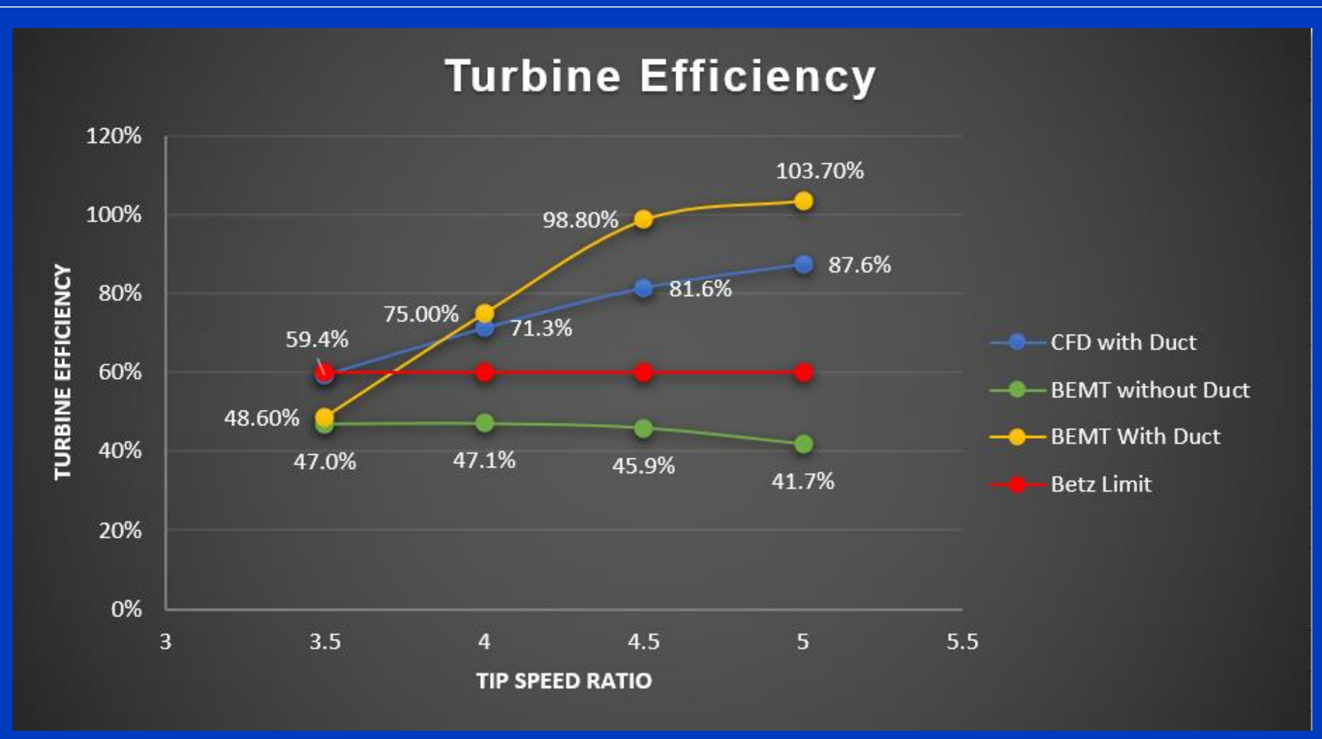


Figure 4: Turbine Efficiency Results

Results

The MATLAB program utilizes BEMT theory to optimize a turbine using the selected airfoil and a duct with an Area Exit / Area Inlet ratio of 1.89. This yielded all the necessary results to complete the design, dictate operating conditions and produce analytical values that are used for comparison with CFD simulation results.

The MATLAB program output data such as: Torque, Thrust and Power. While these parameters are important, efficiency is the most important indication of how well designed and simulated the turbine is. Located above is Figure 4, a graph combining CFD and MATLAB results. The program and Fluent simulations were run at a range of 3 – 5 TSR, with increments of .5 to determine the optimum value. Two versions of the MATLAB program were run: one without a duct and one with a duct. For reference, a turbine with no duct can reach the maximum efficiency set by the Betz limit at 60%. The unducted turbine returned an average efficiency of 45.4%. With the addition of a duct, the turbine averaged 81.53%, with a maximum of 103.7%, far exceeding the Betz limit.

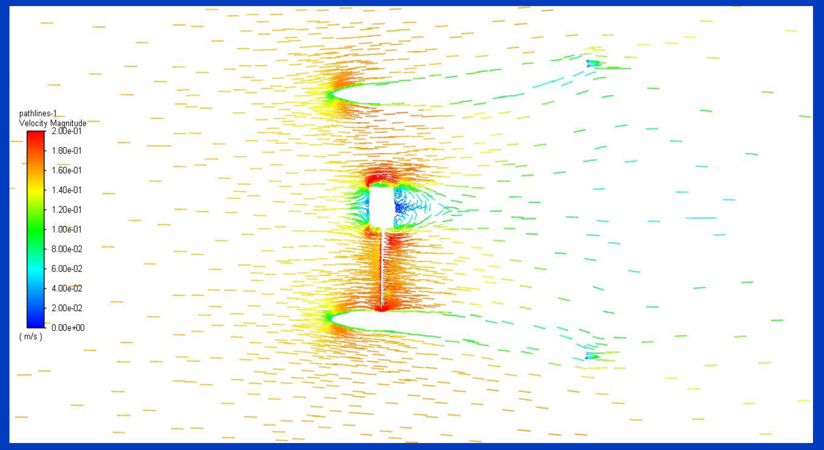
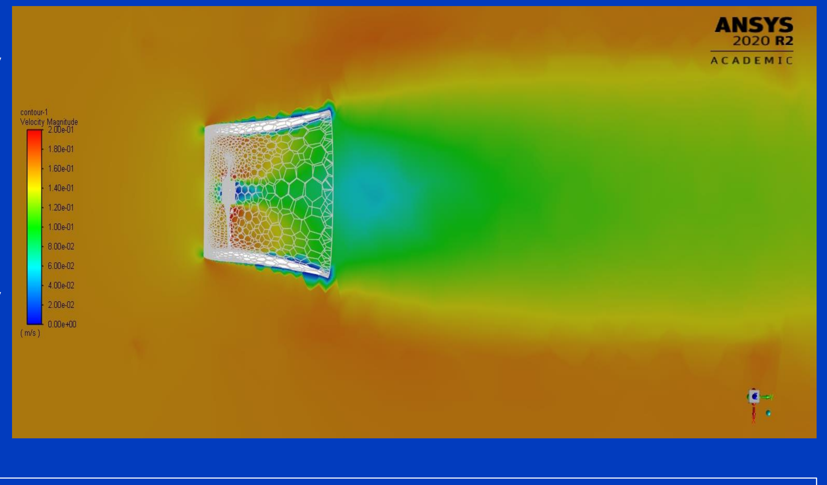


Figure 5: Velocity Vectors

Figure 6: Velocity Contours



Ansys Fluent analysis validates the MATLAB results by running simulations that utilize real life losses and aerodynamic principles. 16 runs were successfully completely and produced quantifiable results. Out of those runs, the best efficiency reached was 87.6% at a TSR of 5. The figures shown demonstrate proper distribution and contours of velocity, further validating that the simulations were done correctly.

Analysis of Results

The researchers wanted to quantify how much of an impact the duct has on turbine performance. The increase in efficiency from no duct to duct was observed in the rest of the performance results, proving the profound effect of the duct, and is therefore a definite necessity for low velocity turbine applications. With the duct in place, the Ansys turbine produced a maximum of 77 Watts. While the numbers between MATLAB and Fluent were rather close, there is still a noteworthy difference between them. A reason for this is that the MATLAB program utilizes the velocity results of the flow going through an empty duct rather than one with a turbine in the middle. The duct alone increases the velocity to ~0.50 m/s and since the power equation cubes the velocity, the increase of the results using this velocity data is rather large. In Ansys, with the turbine in the middle of the duct disrupting the venturi effect, the velocity increased to ~0.30 m/s and produced lower results for each performance parameter.

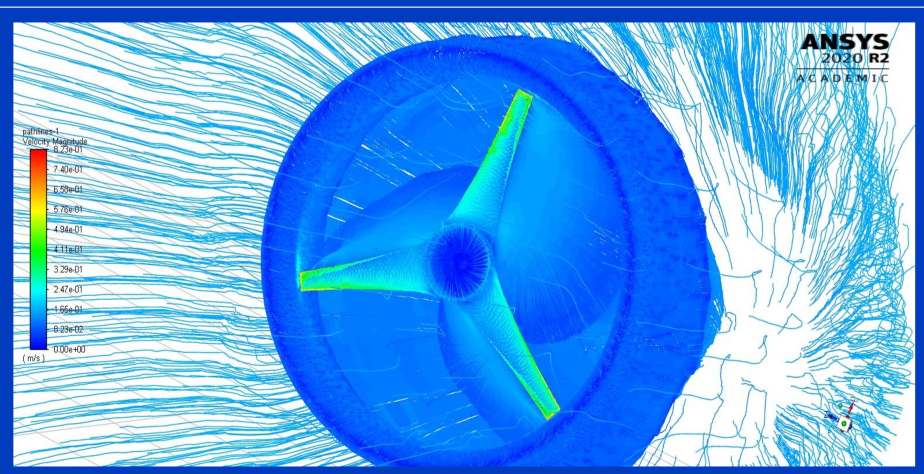


Figure 7: Velocity Streamlines

Conclusion

The research objective of simulating and generating electricity from a hydrokinetic turbine in the water conditions of Puerto Rico was achieved. The turbine exceeded the Betz limit of efficiency with the addition of a duct by reaching an efficiency of 87.6% while running at only 0.16 m/s. It was designed using fundamental propeller-based theories and current research papers about the optimization of ducted hydrokinetic turbines modified for a thoroughly researched airfoil.

Is there a future for hydrokinetic power in Puerto Rico? It is too early to tell. Currently, the biggest hurdle is the limited places that could successfully power a hydrokinetic turbine. Until there is a workaround to this issue, the basic laws of physics limit performance to the results highlighted here. Despite this, based on the results of this research, the researchers believe the future is bright. Puerto Rico has its goal set on generating 100% of its electricity via renewable means by 2050. What is certain is that the beautiful coasts of Puerto Rico can have a part in making that goal a reality in the future.

Future Work

The most prominent limitation in this investigation was the lack of data about the waters around the island. Finding areas with higher velocities could greatly improve the case for implementing this type of turbine, while also taking into consideration other parameters such as tourism, sea life and pollution. Another area of improvement is better CFD analyses. All simulations were done utilizing Ansys Workbench for Students, limiting the number of elements/nodes to 512K. While the team is confident in the results, having higher quality simulations and more accurate results is beneficial. Material study, turbine foundation and electrical component assessment would bring the turbine closer to reality.

References

Allsop, S., Peyrard, C., Thies, P. R., Boulougouris, E., & Harrison, G. P. (2017). Hydrodynamic analysis of a ducted, open centre tidal stream turbine using blade element momentum theory. Ocean Engineering, 141, 531-542. https://doi.org/10.1016/j.oceaneng.2017.06.040

Kempener, R., & Neumann, F. (2014). Tidal energy: Technology brief 3. International Renewable Energy Agency. https://www.irena.org/publications/2014/Jun/Tidal-Energy

Mete Sireli, E. (2014). Design implications of ducts on horizontal axis turbines. The Journal of Ocean Technology, 9(2), 68-90. https://www.thejot.net/article-preview/?show_article_preview=554

Shives, M., & Crawford, C. (2011). Developing an empirical model for ducted tidal turbine performance using numerical simulation results. Journal of Power and Energy, Part A of the Proceedings of the Institution of Mechanical Engineers, 226(1), 112-125. https://doi.org/10.1177/0957650911417958

Acknowledgments

The researchers would like to thank Prof. Jose Pertierra for all the support in every step of this investigation, especially in the MATLAB and Ansys portion of the research. A special thanks to Etienne Lopez for advising us in the creation of this poster. Immense gratitude to the Polytechnic University of Puerto Rico and the URP-HS program for offering this great opportunity to research such interesting topics.