A Validation Study on the Performance of a Horizontal Axis Tidal Turbine (HATT)

Binoe E. Abuan

Department of Mechanical Engineering, College of Engineering
University of the Philippines-Diliman, Philippines

Abstract – A Computational Fluid Dynamics (CFD) model of the Sheffield HATT, a tidal turbine designed to perform better in varying flow velocity, was created in order to predict its performance in unsteady flow using ANSYS FLUENT. Several mesh independence tests were performed to determine a suitable mesh for the problem under study. The final CFD model was used to do simulation of a full tip-speed ratio (TSR) sweep to determine the performance curve of the turbine. It was shown that at Reynolds number, Re=125,000 (which is the same Re at the wind tunnel experiment), the shape and the trend of the performance curve has the same shape when compared to that of the initial Re at 1,350,000. Lower values of CP were obtained which agreed with a separate Blade-Element Momentum study where CP values go down as Re decreases. Brake torque experiments (using spring balance) in the wind tunnel were conducted at Re=135,000 to obtain a power curve that is plotted against the CFD simulation results. It was shown that there is good correspondence between the experimental results and the CFD results which in turn gives more confidence for the numerical data obtained from CFD simulations.

Keywords—Brake torque experiment, CFD, Sheffield HATT, validation

I. INTRODUCTION

Tidal energy is currently gaining attention both from the academe and the industry because of the large potential it is offering to the renewable energy scene. At the center of research and development are tidal extraction devices and horizontal axis tidal turbine (HATT) is the most common. Experimentation and testing of HATT prototypes will require large amount of capital expenses, effort, and time, that is why numerical simulations are being used currently to replace prototyping in energy systems. In HATT systems, BEM and CFD simulations are the most popular. Very informative data and information can be obtained from a single CFD simulation which will cost tons of money when done in experiments. Although numerical simulations have the potential to make energy system testing cheaper and more practical, these tools need to be validated through experiments to gain confidence on the results being obtained.

In this study, a tidal turbine designed to work better in unsteady flow will be analyzed with the help of BEM and CFD simulations. The initial design (true scale) of the turbine and its performance will be presented first. A small-scale version (wind tunnel scale) of the initial design will be used for the validation study against experimental results from a simple brake torque measurement done in the wind tunnel under the same Re as that of the numerical model. This way, if the same setting used for the validation model is applied to the initial CFD simulation, then there will be higher confidence on the results obtained through CFD simulations.

II. LITERATURE REVIEW

Tidal turbine analysis has been presented in both physical and numerical simulations in several literatures already. A series of experiments on tidal turbine performance was conducted in the University of Southampton led by Bahaj and Batten [1, 2]. They looked at the feasibility of using a cavitation tunnel and towing tank for turbine analysis through power analysis and Blade-Element Momentum validation. Wang et al did experiments in a cavitation tunnel used for marine propellers to look at the onset of cavitation in tidal turbines [3]. An experiment using Particle Image Velocimetry (PIV) to analyze the wake formed by a model tidal turbine in an open channel flow was conducted by Chamorro et al [4] while Morris examined solidity, wake recovery, and blade deflection on HATT using a water flume [5]. These literatures show that small scale modelling in different laboratory environments such as cavitation tunnels, towing tanks, open water channels and water flumes can present good data for different tidal turbine parameters being studied. In this paper, an experiment carried out in a wind tunnel at a lower Re is presented to validate the CFD modeling done by the author through power curve comparison.

In terms of numerical simulations, the literatures presented mostly are Blade-Element Momentum (BEM) Method and Reynolds-Averaged Numerical Simulations (RANS). The BEM method has been validated by Batten et al [2] and was compared with their experimental results in reference [1]. BEM has also been integrated into wind and tidal turbine performance and design software like Hassan's Bladed and Tidal Bladed which is validated by the Energy Technological Institute. The author of this paper has validated QBlade, an open source BEM solver, in previous studies. [6, 7]. Malki et al [8] used a coupled BEM-CFD model to address the limitations of the BEM but still have faster simulations than pure CFD analysis. Tidal turbine performance has been simulated in CFD by Milne et al [9,13] while unsteady flow performance is presented in Abuan [10].

III. DESIGN OF THE SHEFFIELD HATT

The Sheffield HATT was designed to achieve a more consistent power curve (power coefficient, CP versus tip speed ratio, TSR) compared to conventional tidal turbines by having a flatter curve near the optimum TSR. Having a high performance at a larger variation of TSR means that the turbine will perform good even if the water flow is varying and unsteady which is the case of the tidal flows under the Philippine water.

The resulting design after a number of iteration is a 2 meter three bladed HATT that uses the NACA 44xx series of airfoil across the span of the blade. The length of the turbine is again referred to the Philippine tidal turbine sites where the channels are shallower and narrower. The airfoil used was recommended for tidal operations by Meyers [11] and was used in previous designs of the Marine Current Turbine's Seaflow project [12]. A detailed design of the Sheffield HATT is presented in Abuan [6,7,10] and the resulting geometry is presented in Table 1 and Figure 1.

The performance of the Sheffield HATT is compared to a tidal turbine validated by Batten et al [2] in their series of experiments regarding tidal turbine hydrodynamics. An initial comparison is presented using Blade-Element Momentum method of the open source software QBlade and is presented in Figure 2. Validation of the said software is presented in detail in Abuan [6,7]. The simulation is for water flow speed of 2 m/s with TSR range from 2 to 10. It can be seen that from the BEM simulations, the Sheffield HATT shows a flat performance curve near the peak TSR although the difference in the peak power coefficient (CP) can be argued to be caused by the limitations of the BEM which does not account for all of the three dimensional effects happening on the blade and is purely based on the lift and drag data of the foils used in the blade.

In an structural response study using Finite Element Analysis (FEA) done by Danao and Abuan

[6,7], it was shown if the material used for the blade is E-glass reinforced epoxy, the Sheffield HATT blade has low stress and deformation levels at 2 m/s flow. Also, it was shown that the blade can withstand typhoon affected flows which is tested at 5 m/s.

Table 1. Sheffield HATT Geometry Specification

Radial Position (m)	Chord Length (m)	Twist (°)	Foil Profile
0.4	0.25	20	NACA 4424
0.6	0.2312	14.5	NACA 4420
0.8	0.2126	11.1	NACA 4418
1.0	0.1938	8.9	NACA 4417
1.2	0.175	7.4	NACA 4416
1.4	0.1562	6.5	NACA 4415
1.6	0.1376	5.9	NACA 4414
1.8	0.1188	5.4	NACA 4413
2.0	0.1	5	NACA 4412

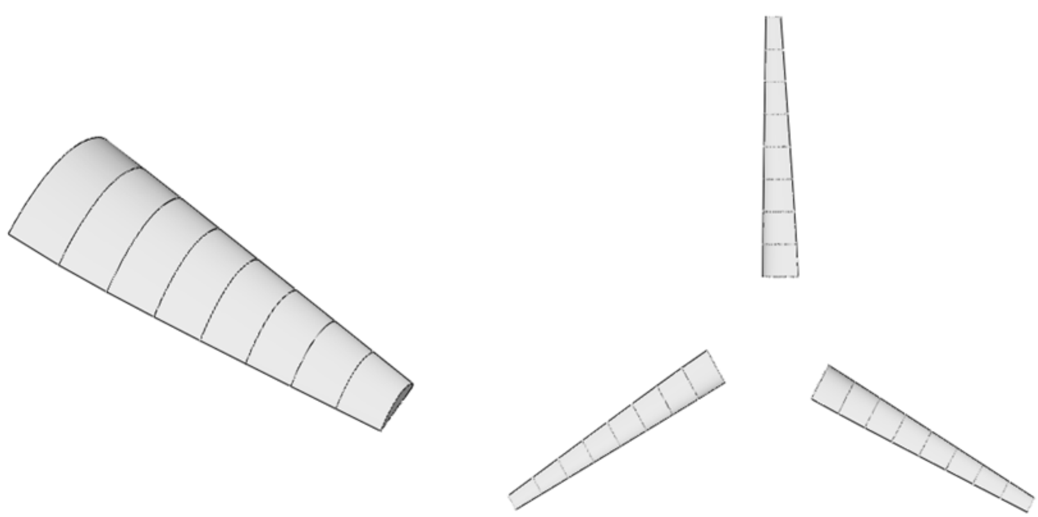


Figure 1. Sheffield HATT model from the turbine building function of QBlade

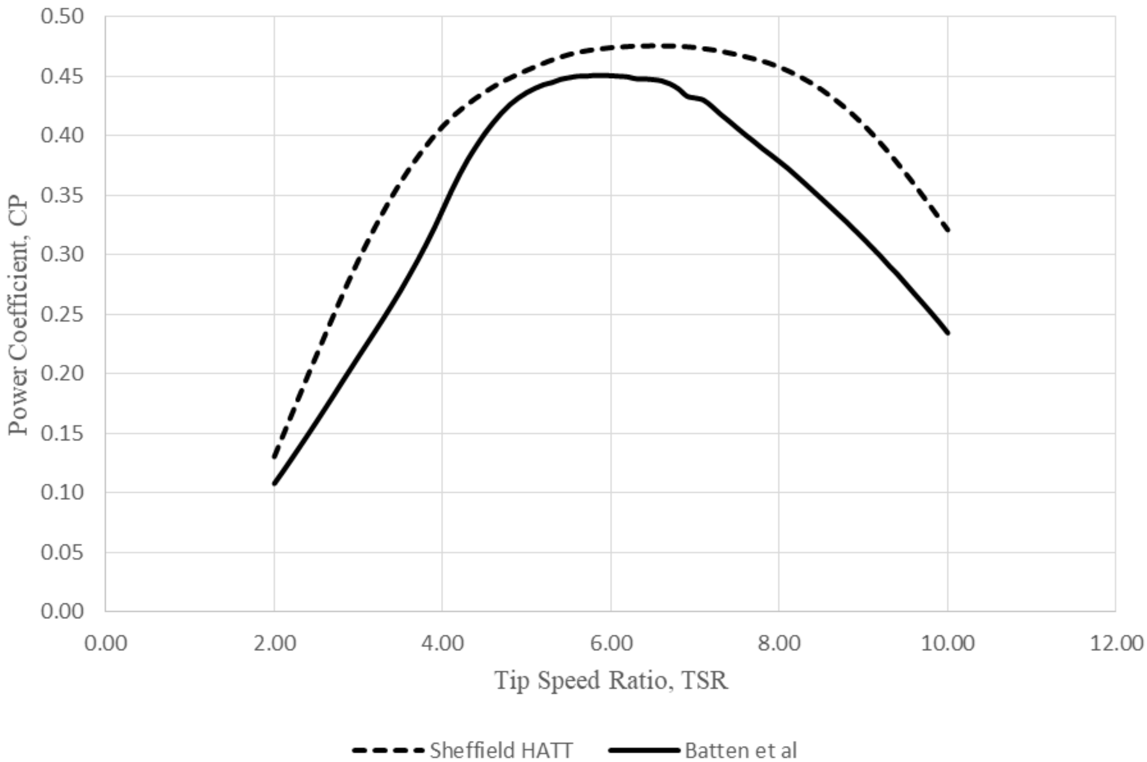


Figure 2 Performance curve comparison of the Sheffield HATT model.

IV. CFD MODELLING OF THE SHEFFIELD HATT

A 3D CFD model for the Sheffield HATT was created using ANSYS-ICEM. Unstructured mesh with generated prism layers near the blade is the meshing technique implemented as shown in Figure 3. A y+ value greater than 30 was targeted to have the first cell height in the fully turbulent region which is necessary for Standard Wall Functions (SWF) of ANSYS CFD. The Sheffield HATT is enclosed in a rotating mesh 2.5 times its diameter it was appended into a stationary fluid domain measuring 5D by 5D by 10D, where D is the diameter of the Sheffield HATT as shown in Figure 4 and 5. The domain extent consideration was based on the University of Sheffield's wind tunnel which will be used for a validation of the turbine performance at low Re.

Before using the model, series of numerical tests were conducted and this is explained in detail in Abuan [10]. This includes a mesh independence study where a final mesh with a target of 300 cells around the blade at the 75% span was selected – corresponding to a total cell count of 4,259,402 and a computational time of 20 hours running at 48 cores. The selected mesh was also tested for boundary size effect, and it was seen that when the boundary is extended twice its size, there is negligible difference in performance. Both study presented are simulated using ANSYS FLUENT at a water flow speed of 2 m/s. The selected mesh through the tests is the one used for all the proceeding simulations for this study.

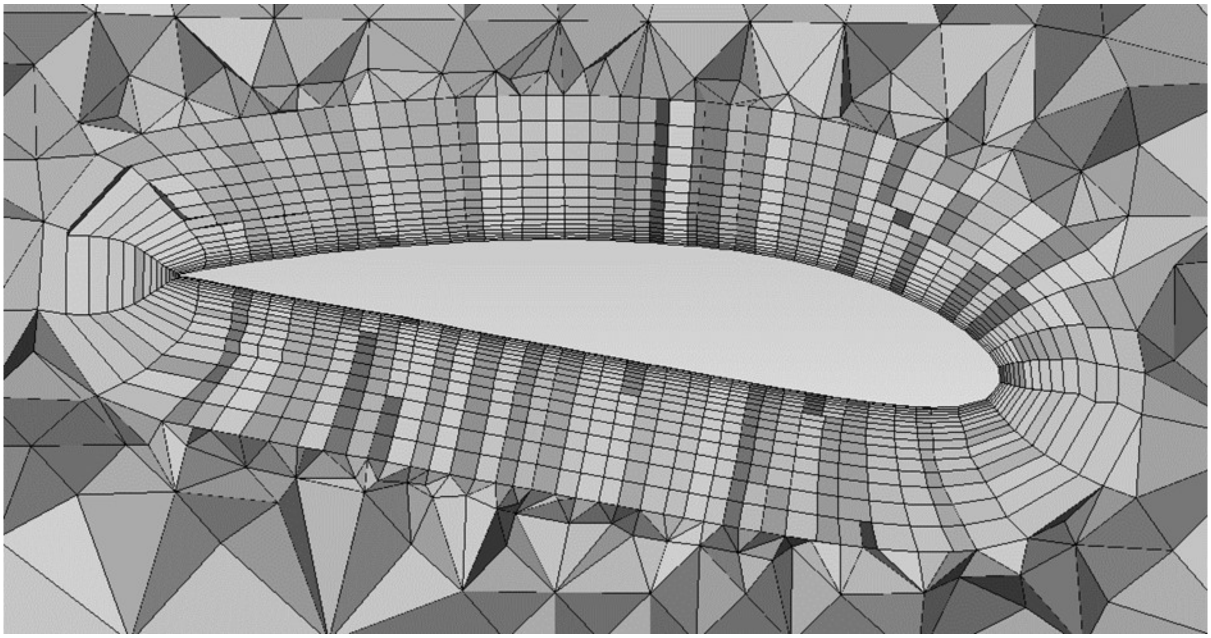


Figure 3. Cut plane mesh for the Sheffield HATT at 75% span of the blade

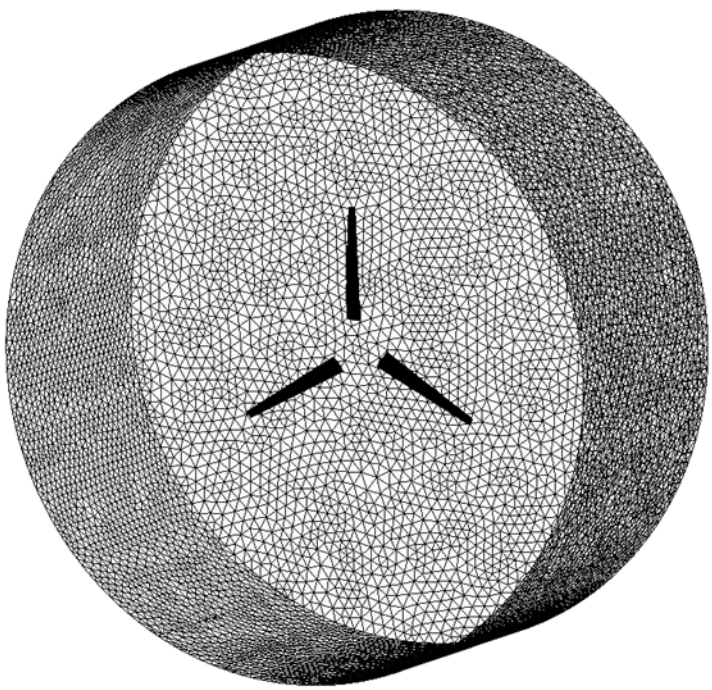


Figure 4. The rotational domain for the Sheffield HATT showing the geometry with mesh

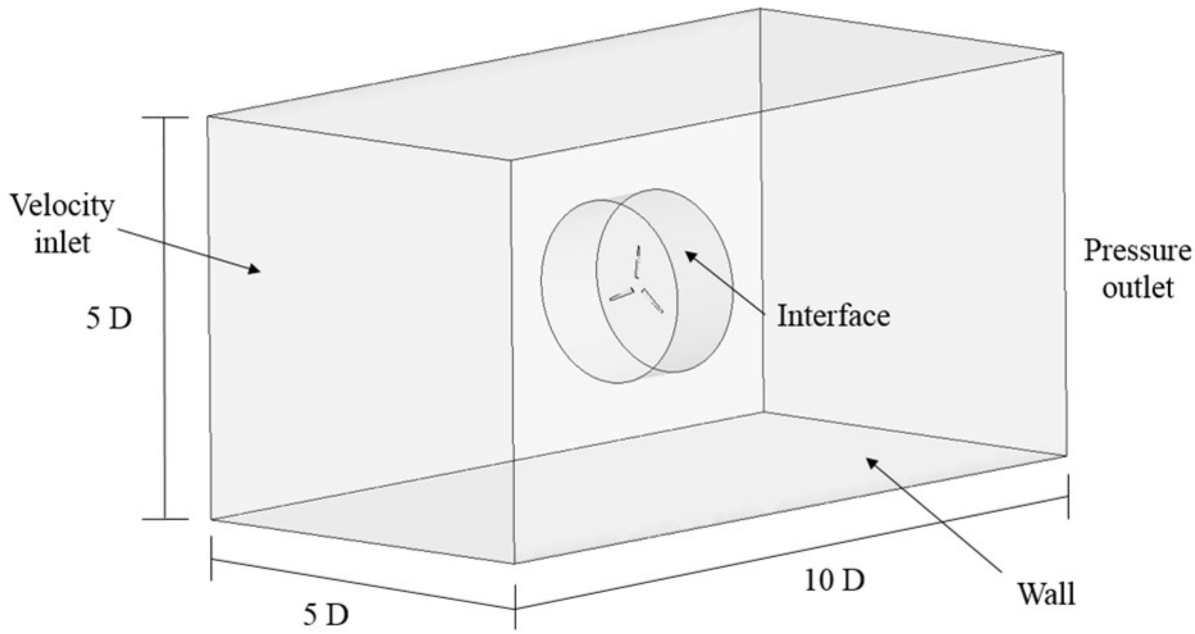


Figure 5. Outer mesh computational domain showing the boundary conditions used in the CFD model (true scale)

V. TURBULENCE MODEL STUDY AND STEADY FLOW SIMULATIONS

Two turbulence models were used to simulate the steady flow performance of the Sheffield HATT. The selected turbulence models are the most common being used for turbine analysis. The first model used was the k-ε RNG model as it is one of the most robust models available in ANSYS FLUENT, it is known to converge faster than other turbulence models but has some problems capturing swirls and flow separation at its early stages. The other model used is the k-ω SST which is an enhanced version of the k-ε model but is known to converge than other models. It is necessary to determine if a faster model like k-ε can capture the physics of the flow for the Sheffield HATT or a slower but enhanced model is needed.

Steady state simulations are conducted for a water flow of 2 m/s corresponding to a Re=1,350,000 (at the 75% span of the blade) for TSR's between 2 and 10. Time-step was set to be one degree of rotation. Residuals per simulation were monitored and convergence was set at 5e-5 for continuity residuals which was usually achieved after 8 turbine revolutions. CP is computed (eq. 1) for various TSR and the resulting curves are presented with the QBlade's BEM result for the same operating condition. Plots of the performance results are presented in Figure 6.

$$Power\ Coefficient, CP = {NT_{blade}\omega} \Big/ {\tfrac{1}{2}\rho A {V_{water}}^{3}} \quad \text{Eq. 1}$$

Where
- N =number of blades
- T_{blade} =blade torque
- ω =rotational speed
- ρ =density of fluid
- A =swept area by the blades
- V_{water} =water flow velocity

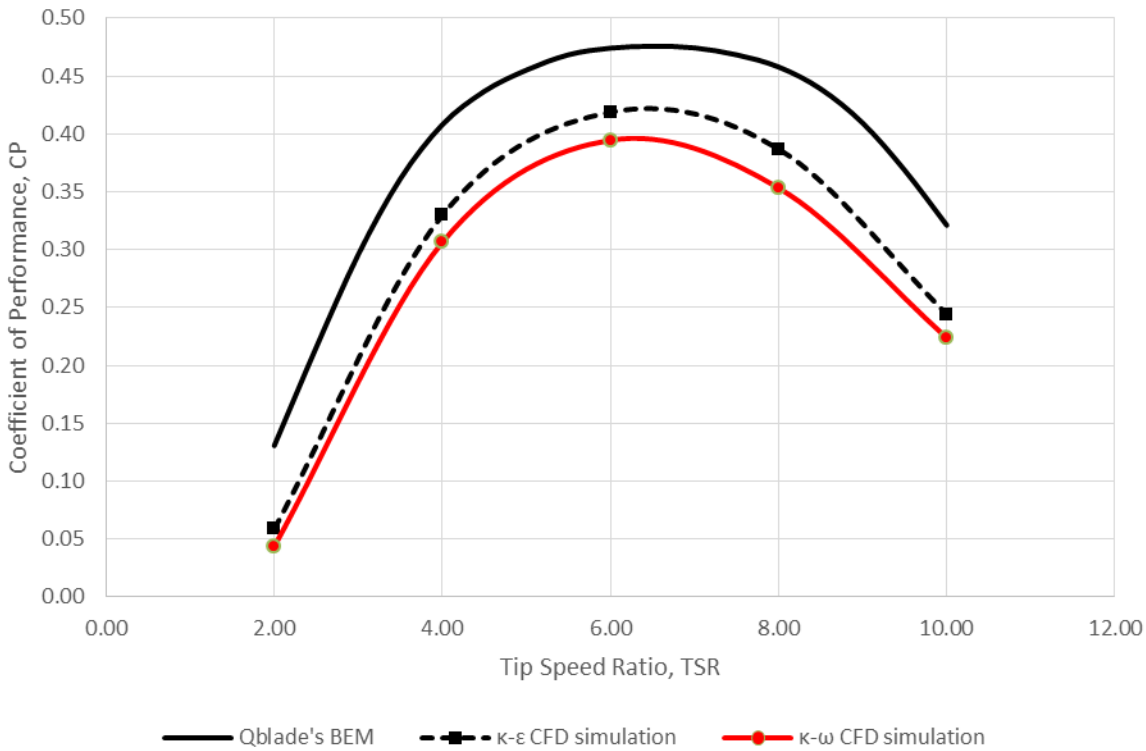


Figure 6. Power curve comparison between BEM, k-ε RNG, and k-ω SST simulations

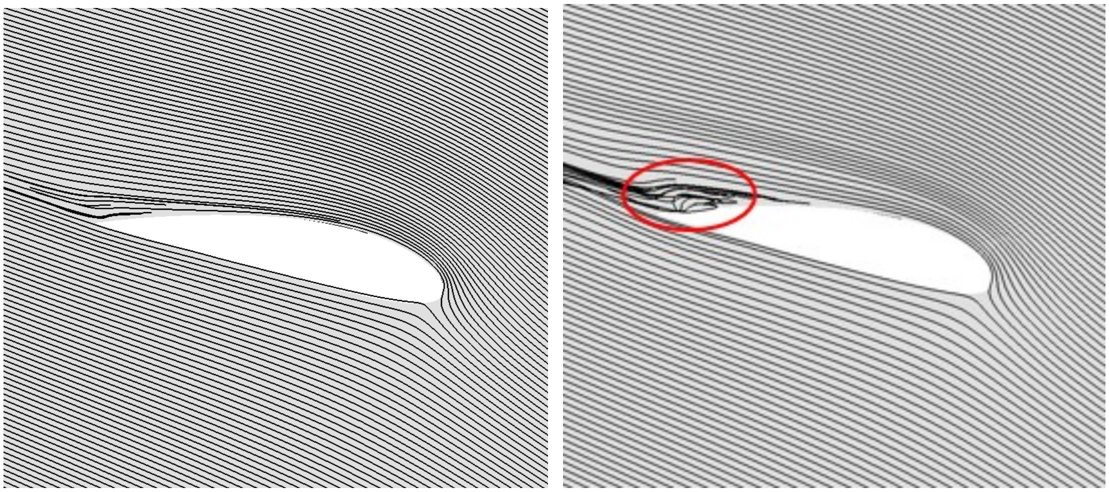
It can be seen that there is good correspondence among the three power curves for the three different numerical simulations. Values at the optimum TSR (TSR=6) is presented in Table 2 and it is shown that lower CP values are obtained for the CFD simulations when compared to BEM. This can be attributed to the fact that BEM simulation only depends on the lift and drag data supplied to its algorithm while CFD simulations accounts for all of the fluid flow effects happening on the flow.

Table 2. Coefficient of performance at optimum TSR (TSR=6)

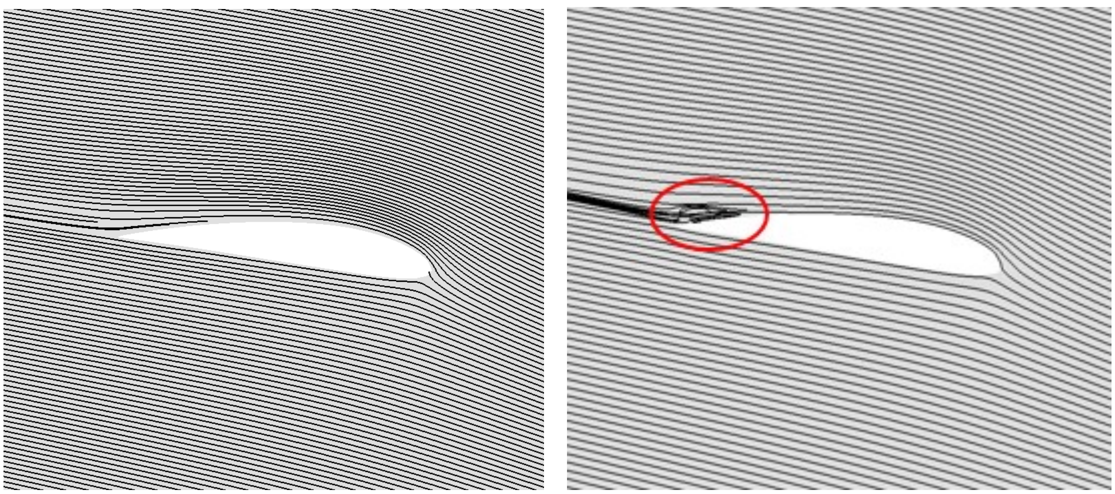
Simulation	Coefficient of Performance (CP)
BEM	0.474
k-ε RNG	0.419
k-ω SST	0.395

Lower TSR corresponds to higher angle of attack (AoA) which means higher lift should be produce but the power curve shows a lower CP at low TSR. One of the main reasons for this phenomenon is the presence of flow separation which resulted from having an AoA beyond the critical AoA resulting to stall. If flow separation is present, drag will also increase drastically which will make the lift to drag ratio lower. Since CP is dependent on Cl/Cd, when the ratio decreases, CP also decreases. Flow separation can be easily determined through flow visualizations of streamlines passing through the blade at different locations. A case study at TSR=4 is presented where Isolated flow visualizations at the 25% and 75% span of the blade is shown in Figure 7 for both turbulence models.

No flow separation was captured by the k-ε RNG model while both locations showed flow separations for the k-ω SST model. Since flow separation like this is critical to explain the physics of the flow around the blade, the k-ω SST model was selected to be used for the proceeding simulations in this study. The hydrodynamic of flow at high TSR will be presented later with the validation study.



25%R



75%R

Figure 7. Isolated flow visualizations for 25%R and 75%R at TSR=4 with k-ε RNG (left) and k-ω SST (right)

VI. RE EFFECT ON THE PERFORMANCE CURVE

The validation study using the wind tunnel will require a lower Re because of the limitations of the wind tunnel hence the effect of Re to the performance of the Sheffield HATT was investigated using QBlade's BEM function. Airfoil performance at different Re was generated and was used for the BEM simulation. Figure 8 shows the performance curve of the Sheffield HATT at various Reynolds number.

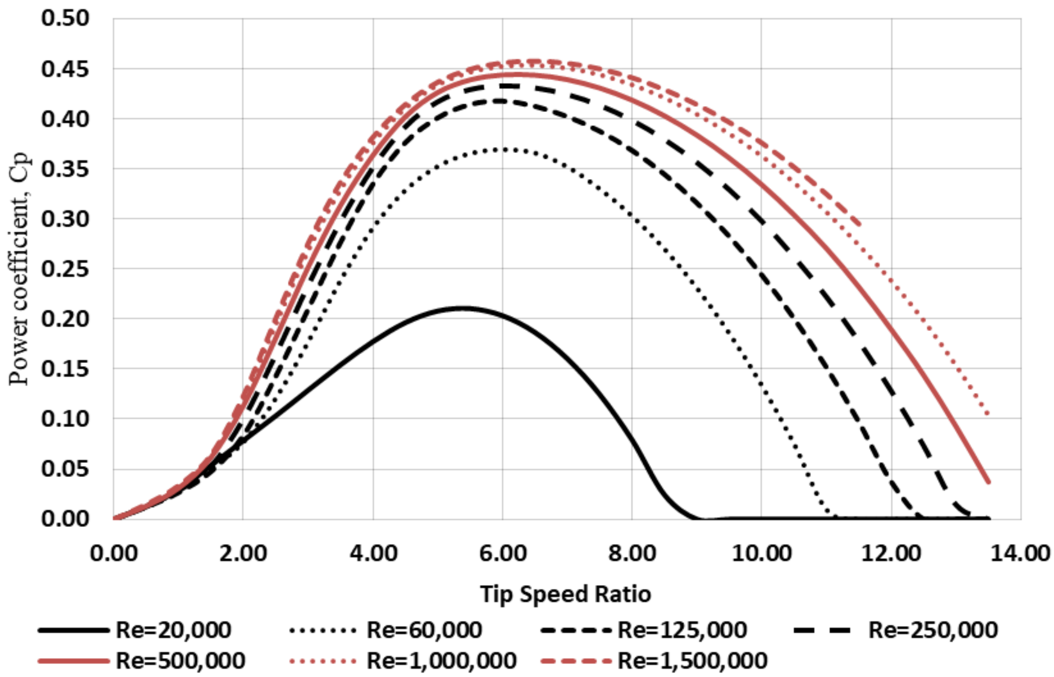


Figure 8. Sheffield HATT performance at different Reynolds number

It can be seen from Figure 8 that at Re less than 100,000, a small difference in Re results to a big change in the power curve. The difference between the power curves at Re=20,000 and Re=60,000 is the biggest despite having the smallest variation in Re number. This effect was minimized once Re became greater than 100,000. Power curves at Re=1,000,000 and Re=1,500,000 shows very minimal difference for a difference of 500,000 in Re number. These observations suggests that the effect of Re becomes larger as Re decreases.

The Sheffield HATT was also simulated at different Reynolds number using FLUENT. Simulations were conducted at optimum TSR (TSR=6) and results are shown in Table 3 The difference in CP decreases as Reynolds number decreases which supports the observation stated above. The effect of Re is important as it gives the idea that even though the Sheffield HATT is running at a fully turbulent Re of 1,350,000, an experimental simulation conducted at Re=125,000 will provide us the general trend of the performance curve.

Table 3. Sheffield HATT CP at various Re

Reynolds Number	CP
125,000	0.33830
500,000	0.37789
1,000,000	0.38997
1,500,000	0.41881

VII. WIND TUNNEL SCALE CFD MODEL AND THE EXPERIMENTAL TURBINE SPECIFICATIONS

An experimental validation for the steady flow performance of the Sheffield HATT was conducted in the wind tunnel laboratory of the University of Sheffield. Because of the size limitation of the wind tunnel, a scaled-down model of the Sheffield HATT was created with geometry specification presented in Table 4. The wind tunnel has a test section with a dimension of 2.4 m by 1.2 m x 1.2 m with a maximum wind velocity of 25 m/s which translates to a maximum Re of around 200,000. Due to safety reasons, the experiment was decided to be executed at a lower velocity which converts to Re=125,000.

Table 4. Sheffield Hatt Geometry Specifications (wind Tunnel Scale)

Radial Position (m)	Chord Length (m)	Twist (°)	Foil Profile
0.0668	0.04175	20	NACA 4424
0.1002	0.03861	14.5	NACA 4420
0.1336	0.03555	11.1	NACA 4418
0.167	0.03236	8.9	NACA 4417
0.2004	0.03	7.4	NACA 4416
0.2338	0.02609	6.5	NACA 4415
0.2672	0.023	5.9	NACA 4414
0.3006	0.01984	5.4	NACA 4413
0.334	0.0167	5	NACA 4412

A scaled-down mesh of the Sheffield HATT model was also created using ANSYS-ICEM to match the geometry restriction of the new model. The size of this turbine matched the size of the experimental turbine. The computational domain was also sized down to the size of the wind tunnel's test section (2.4 m x 1.2 m x 1.2 m). The new mesh was then simulated at Re=125,000 in FLUENT to match the experimental Reynolds number. Reynolds number matching was done so that a validation study can be executed even though the fluid used is different for the CFD simulation and the corresponding experiment. All of the previous settings selected in the mesh studies are applied for the smaller mesh.

The CP of the scaled-down model was recorded for various TSR to complete the power curve which was compared again to the corresponding BEM result gathered from QBlade. The comparison plot is shown in Figure 9. The results show an expected power curve lower as compared to that at high Reynolds number. Maximum CP for the BEM simulation is at 0.418 while a CP of 0.338 is observed for the CFD simulation, both occurs at the optimum TSR=6.This observation agrees to the QBlade BEM's simulation for the effect of Reynolds number to the HATT's performance whereas a lower power curve is expected.

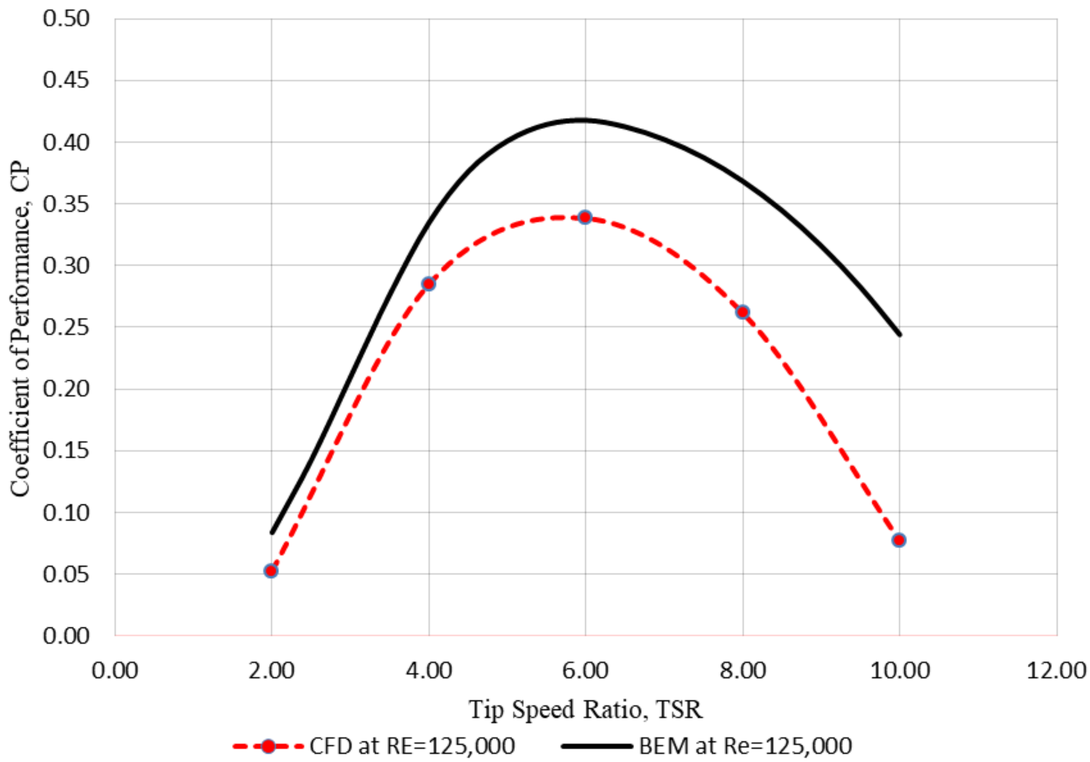


Figure 9. Power curve results for the scaled-down mesh (wind tunnel scale) simulated in CFD and QBlade's BEM

VIII. EXPERIMENTAL VALIDATION OF THE SHEFFIELD HATT'S PERFORMANCE CURVE

An experimental validation was done for the scaled down model of the Sheffield HATT in the University of Sheffield's Wind Tunnel Laboratory. The idea is for the experiment to match the Reynolds number of the CFD model, which is Re=125,000 at a water velocity of 1 m/s, to serve as an experimental validation of the mesh. A previous CFD simulation was done to prove that flows at the same Reynolds number, though different fluid material, have very comparable results in terms of performance in Abuan [6]. The scaled down HATT mesh was simulated at a wind velocity of 15 m/s to have a Re=125,000 at TSR=6. The resulting CP for that simulation was 0.3403 which has a difference of 0.59% when compared to the CP at that simulated using water which is at 0.3383.

The swept area of the scaled down model corresponds to a blockage ratio of about 20% which is based on previous blockage ratio testing in the Wind Tunnel (Edwards et al [14]). The blockage ratio was maintained at this value to assure that the flow rate throughout the test section has a very minimal variation if not constant as confirmed on those previous studies.

IX. EXPERIMENTAL METHODOLOGY AND RESULTS

The first thing to do in the experiment is to measure the velocity of the wind flow in the wind tunnel. This is measured using two pitot tubes as shown in Figure 10. The pitot tubes are connected to a pressure transducer which provides values of static and dynamic pressures. Velocity can be solved using the acquired pressures and applying the Bernoulli's equation; $p + ½ \rho V^2 = \text{constant}$ (the term ρgh was neglected because there is no height difference inside the wind tunnel). The density used for the computation is the current density of air inside the room where the air temperature was maintained to be constant throughout the whole experiment. The Reynolds number and the available wind power will be both dependent on the velocity measured.

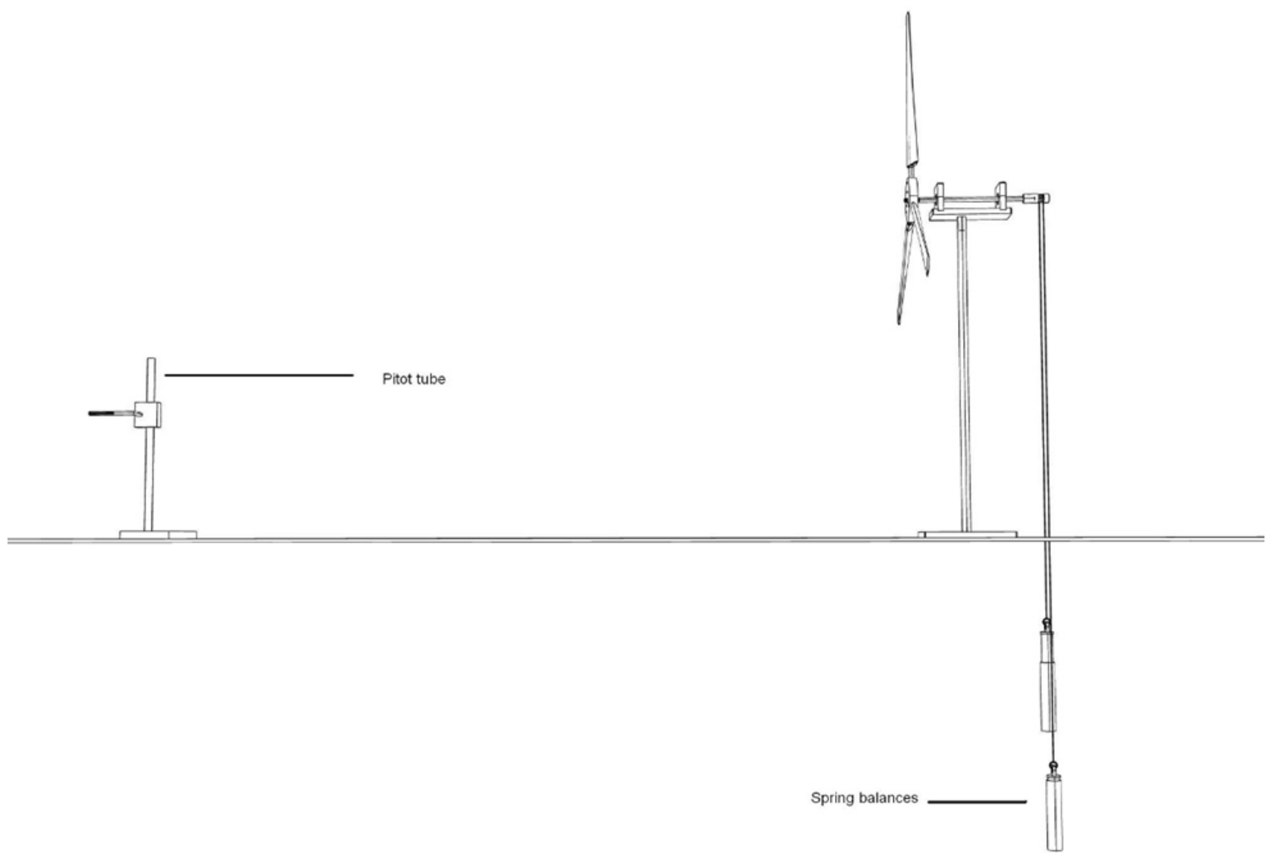


Figure 10. Experimental Set-up for the Steady-state Validation Experiment

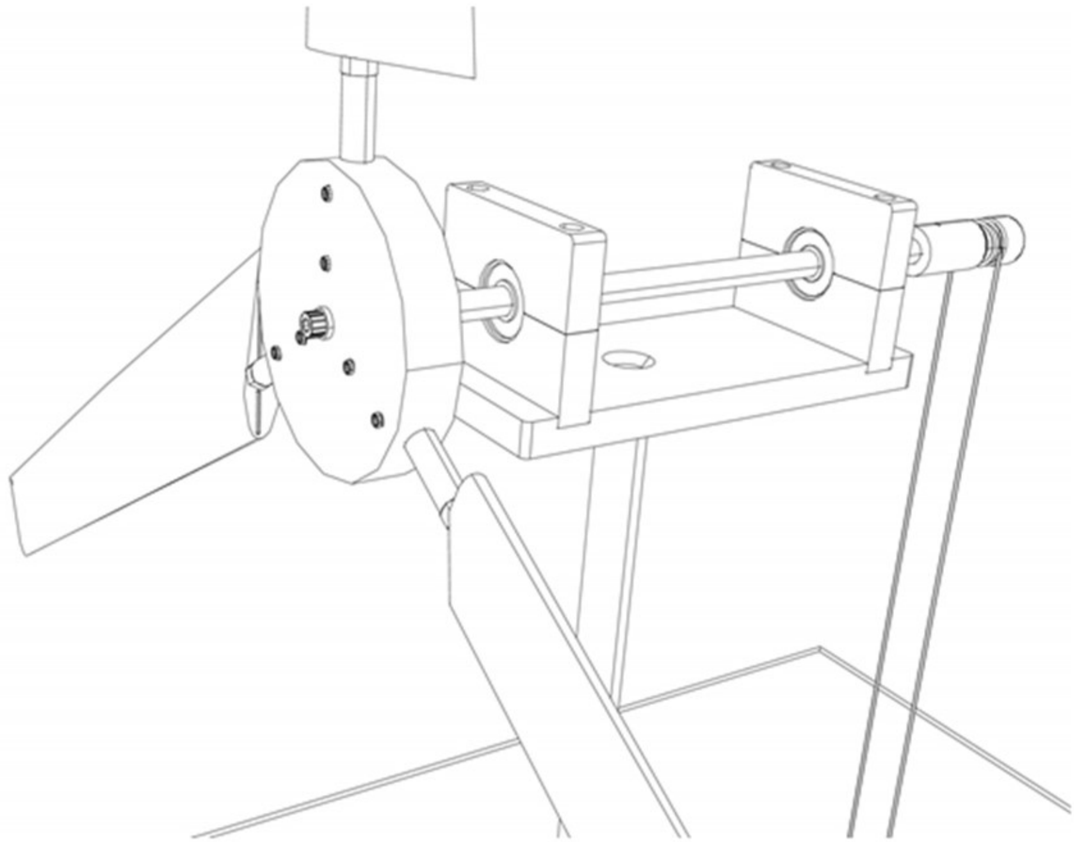


Figure 11. A close-up image of the experimental set up showing the turbine with the wrapped metal string at the end

To compute for the power coefficient, the power generated by the turbine should be computed. This power is dependent on the torque produced by the turbine and the rotational speed of the rotor; $P_{extracted} = T\omega$, where T is the torque in N-m and ω is the rotational speed in rad/s.

The rotational speed of the turbine was measured using a rotational speed meter (RSM) attached to the tower of the turbine. A reflective surface was attached at the back of one of the blades, this surface will reflect the LASER that the RSM is emitting. The RSM will then count the signals and translate it into rotational speed displayed in the RSM's screen.

The torque is measured using a simple torque brake mechanism composed of a metal string wrapped once at the end shaft of the rotor (Figure 11) with ends attached to spring balances, as shown in Figure 10. When the spinning rotor reaches equilibrium (whereas the rotational speed in the rotameter is constant), the readings in the spring balances can be taken. The rotational speed of the rotor can be controlled by moving the spring balances up and down thus controlling the applied load on the system. The difference between the two spring balances multiplied by the radius of the shaft where it is wrapped will give the Torque that can be extracted from the rotor.

There are some limitations experienced while doing the experiment, the first one is the rotational speed and TSR limitation. Because the torque is measured using a simple brake torque mechanism, it is hard to control slower rotational speed. When the experimenter tried to lower rotational speed, the metal string will apply too much brake force halting the turbine.

Three separate experiments are conducted. The average of the three is presented in Table 5 which shows that the experiment only covered the TSR range from 6.5 to 10.5 which is the latter half of the power curve. The CP data points from the experiment are superimposed with the k-ω SST CFD results at low Re. It can be seen that although the resulting data points have lower values for their corresponding TSR's, the general trend of the plot shows correspondence to that of the CFD simulations as shown in Figure 12. This will give confidence to the numerical simulation results as it shows validation with experimental data. Since both CFD settings were used for the initial simulation and the validation simulation, higher confidence can now be given to the CFD results. Lower values were measured in the experiment, for example, a CP of 0.217 was measured in the experiment while a corresponding 0.327 was presented in the CFD simulation. The only data points that overlapped with each other is at TSR = 10. The difference in the performance can be attributed to friction in the bearings, errors in the metal strings and fluctuation in the rotational speed while reading the spring balances which results to lower torque measured.

As the TSR increases, the AoA's at different sections of the blade decreases. Low AoA will mean low performance curve shows a decrease in CP as TSR increases from TSR=6 (peak TSR) to TSR =10.5. Low AoA corresponds to low lift generated in the blades, drag on the other hand is also low and will not change significantly as the flow is fully attached. It will follow that at TSR=6.5 and above, the CP will be mainly dependent on the lift production and since lift is constantly decreasing, the CP will also decrease as shown in both the experimental results and the CFD simulations.

Table 5. Experimental Results using the Brake Torque Method through Spring Balances

TSR	w (RPM)	rad (rad/s)	delta (spring balance)	Torque, N-m	Power extracted, W	CP
10.5	2100	219.91	0.045	0.008	1.845	0.025
10	2000	209.43	0.14	0.026	5.465	0.074
9	1800	188.49	0.17	0.031	5.973	0.081
8.5	1700	178.02	0.24	0.045	7.964	0.108
8	1600	167.55	0.385	0.072	12.024	0.163
7	1400	146.61	0.575	0.107	15.712	0.213
6.5	1300	136.14	0.63	0.117	15.986	0.217

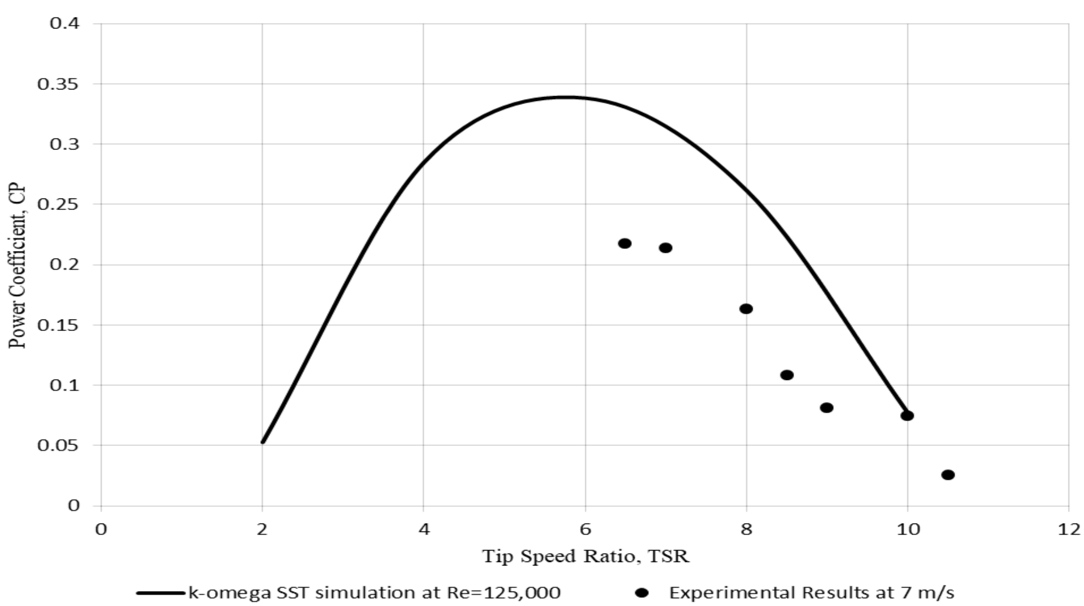


Figure 12. Experimental Validation of the CFD simulation at Re=125,000

X. CONCLUSION

A newly designed tidal turbine called the Sheffield HATT was created with the aim of performing better even when the water flow velocity is varying (which is the case of tidal streams under the Philippine waters) This target design was achieved as was presented in the BEM and CFD simulation results of the final model mesh. Certain mesh studies were conducted before selecting the final mesh. A turbulence model comparison between k-ε RNG and k-ω SST simulation results is presented with the results being k-ε RNG unable to predict the start of separation in the low TSR region. Due to this, k-ω SST was chosen to be used for the rest of the study.

The effect of Re to the performance curve of the Sheffield HATT geometry was presented and it was shown that as Re decreases, the effect of Re in the performance increases. It was justified that for an experiment done at Re=125,000 in the wind tunnel, the trend of the performance curve will still be captured although it is expected to be lower than the initial Sheffield HATT performance curveA scaled-down model of the Sheffield HATT was created to match the dimension of the wind tunnel in the University of Sheffield with a target Re=125,000. Experiments using the brake torque methodology (with the use of spring balance) were conducted matching the CFD simulation. Only the second half of the performance curve was captured in the experiment because of the limitation of having too slow rotational speed that eventually results to halting the turbine. It was shown that there is a good correspondence between the CFD simulation and the experimental results which in turn provide more confidence in the CFD simulation results.

REFERENCES

[1] Bahaj, A., Batten, W., & McCann, J. (2007). Experimental verifications of numerical predictions for the hydrodynamic performance of horizontal axis marine current turbines. Renewable energy, 2479-2490.
[2] Batten, W., Bahaj, A., Molland, A., & Chaplin, J. (2007). Experimentally Validated Numerical Method for the Hydrodynamic Design of Horizontal Axis Tidal Turbines. Ocean Engineering, 1013-1020.
[3] Wang, D., Atlar, M., & Sampson, R. (2007). An experimental investigation on cavitation, noise, and slipstream characteristics of ocean stream turbines. Part A: Journal of Power and Energy, 219-231.
[4] Chamorro, L., Hill, C., Morton, S., & Ellis, C. (2013). On the interaction between a turbulent open channel flow and an axial-flow turbine. Journal of Fluid Mechanics, 658-670.
[5] Morris, C. (2015). Influence of Solidity on the Performance, Swirl, Characteristics, Wake Recovery and Blade Deflection of a Horizontal Axis Tidal Turbine. PhD Thesis. Cardiff University.
[6] Danao, L., Abuan, B. and Howell, R. (2016) Design Analysis of a Horizontal Axis Tidal Turbine, 2nd Asian Wave and Tidal Conference. Marina Bay Sands, Singapore.
[7] Abuan, B. and Howell, R. (2016). Effect of Idealised Unsteady Flow to the Performance of Horizontal Axis Tidal Turbine, 2nd Asian Wave and Tidal Conference. Marina Bay Sands, Singapore.
[8] Malki, R., Williams, A., Croft, T., & Masters, I. (2013). A coupled blade element momentum - Computational fluid dynamics model for evaluation of tidal stream turbine performance. Applied mathematical Modelling, 3006-3020.
[9] Milne, I., Day, A., Sharma, R., & Flay, R. (2012). Blade loads on tidal turbines in planar oscillatory flow. Ocean Engineering, 163-174.
[10] Abuan, B. and Howell, R. (2019). The Performance and Hydrodynamics in unsteady flow of a horizontal axis tidal turbine, Renewable Energy, Vol. 133, pp 1338-1351
[11] Myers, L. (2005). Operational parameters of horizontal axis marine current turbines. PhD Thesis. University of Southampton.
[12] Marine Current Turbines. (2017, january). Seagen Technology. Retrieved from Marine Current Turbines: http://www.marineturbines.com/Seagen-Technology
[13] Milne, I., Day, A., Sharma, R., & Flay , R. (2016). The characterisation of the hydrodynamic loads on tidal turbines due to turbulence. Renewable and Sustainable Energy.
[14] Edwards JM, Danao LA & Howell RJ (2015) PIV measurements and CFD simulation of the performance and flow physics and of a small-scale vertical axis wind turbine. Wind Energy, 18(2), 201-217