10th ANKARA INTERNATIONAL AEROSPACE CONFERENCE AIAC-2019-014
18-20 September 2019 - METU, Ankara TURKEY

DESIGN AND 3D ANALYSIS OF WELLS WAVE TURBINE

Khurram Rehman[1] , Kanza Ramzan[2], and Haroon Javed [3]
Institute of Space Technology,
Islamabad, Pakistan

ABSTRACT

Wells turbine is a self-rectifying air turbine. Self-rectifying means that the blade of the turbine will rotate in a single direction irrespective of the direction of the incoming airflow. The overall configuration of wells turbine is a simple not complex, thus can be used easily to conserve energy. This study is made using CFD i.e. computational fluid dynamics techniques on a 3-dimensional blade made of symmetric airfoils. For this purpose, a single optimized airfoil i.e. NACA 0015 is used for blade profile. The characteristics are analyzed and parameters such as torque coefficient C_T, pressure coefficient and efficiency is calculated and compare with the benchmark results at first and then calculated for the newly designed blade with improve characteristics for different cases. Numerical results show that the turbine characteristics can be increased using 3-dimensional blade as the early flow separation is delayed in this case. The peak efficiency in this case following the benchmark was found to be around 51 percent which was 2 percent higher than the existing 3-dimensional blade turbine. This efficiency will further increase by changing tip clearance and flow coefficient if further research is carries out on it. As a result, it is seen that efficiency of the turbine can be improved using new design.

Keywords— self-rectifying; optimized; flow separation; efficiency

INTRODUCTION

Wells Turbine is a type of an air turbine. It is used in oscillating water columns to convert wave energy in to other usable forms. The principle of working of wells turbine lies in the fact that the rotor of the turbine rotates in a single direction irrespective of the direction of oscillating water column. Hence, it can be characterized as self-rectifying. Wave energy from oscillating air column due to motion of waves is used to drive the turbine. Wells turbine is located mostly on oceans, as it provides necessary energy from the motion to air column above it to drive the turbine. Whereas, after passing through the blades, air waves move out from the other duct above the blades. Thus it acts like a non-return valve in order to rectify the oscillating airflow, working together with the turbine.

For the ocean wave turbine there are ways, applying which we can improve the characteristics of turbine. The characteristics can be improved by use of guide vanes, changing tip clearance, modified blade geometry, and flow coefficient. Using modified blade geometry, efficiency of turbine can be increased by moving the maximum thickness location.

[1] Assit. Prof. in Department of Aerospace, Email: khurram.rehman@ist.edu.pk
[2] Researcher in Department of Aerospace, Email: kanza14@IST.edu.pk
[3] Researcher in Department of Aerospace, Email: haroon76@gmail.com

The optimized airfoil NACA0015 generated by the research [S. Shaaban,2017] is used to generate the 3D geometry. The rotor with eight optimized blades is used for the analysis. Numerical simulations are carried out in the CFD using ANSYS fluent, whereas, the process of generating grids is done in Gambit. 3D simulations are performed under steady flow conditions in order to find the optimum efficiency. Whereas, the numeric results calculated in terms of dimensionless coefficients are compared with the one from the earlier research [S.Shaaban,2017].

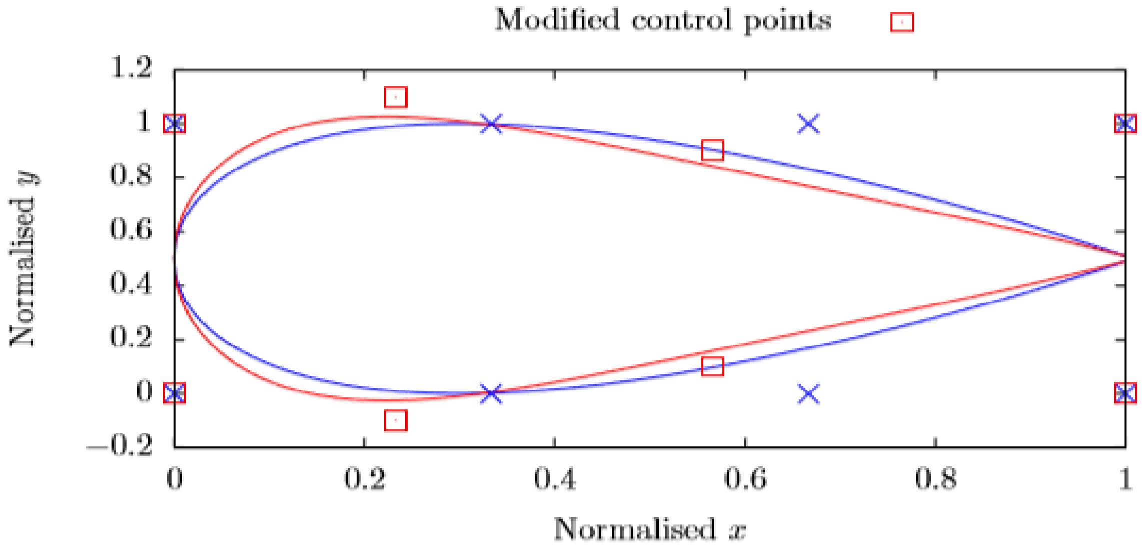

Figure 1:Adjustment of control points – optimized airfoil – existing airfoil

METHODOLGY

Geometry

The design of 3-dimensional blade is using symmetric airfoil with constant chord length. Airfoil coordinates were imported in CATIA and the eight blade rotor was constructed with the parameters given in following Table 1.

Table 1: Geometry parameters of improved turbine

Parameters	Values
Rotor tip diameter	590 mm
Rotor hub diameter	400 mm
Tip clearance	1 mm
Chord length	125 mm
No. of blades	8
Rotational speed	800 rpm

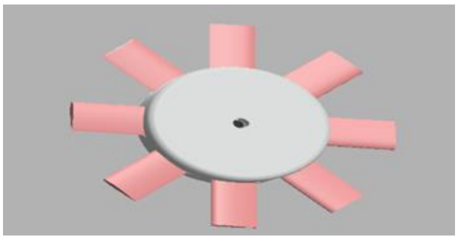

Figure 2 Complete Rotor

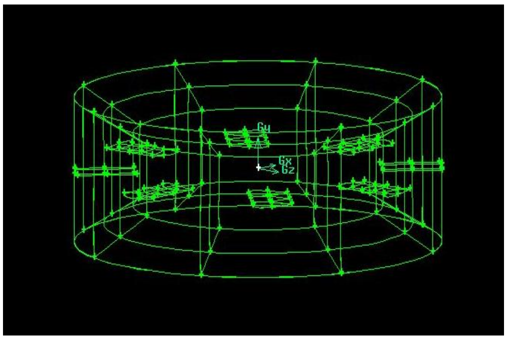

Figure 3 Complete Flow Domain

Meshing

For the meshing purpose, Gambit 2.4.6 is used. For this all the edges were selected one by one. Elements type was selected as quad whereas mesh type as pave. Interval size was selected as 0.02 and interval count was selected as 60 for the side pillars, 120 for the top pillars and 100 for the below pillars. Similarly, interval count for the airfoils was 80.
After edge meshing, the next step is face meshing. For this purpose similar method is followed as in edge meshing i.e. selecting all the faces and applying similar interval count for similar size faces. Type of meshing was selected as pave where elements were selected as Quad. For the face mesh grid cluster gradually thin out while moving away from the edges. Whereas, spacing was 0.02 and interval count was selected as in the edge meshing with the same number.
Moving on to the last step volume mesh was done, using a single volume of the whole blade keeping the same type of mesh. Figure of which is as under. After volume mesh, the quality of mesh was examined, from the summary it was clear that the quality of mesh was found to be satisfactory. As no elements were in the region from 0.9 to 1, hence it was near to ideal mesh quality. Figure 4 below describes the edge, face and volume meshing.

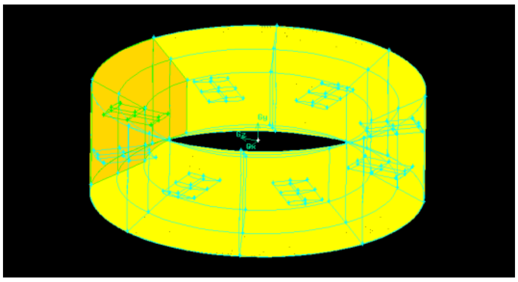

Figure 4 Volume mesh on a Complete Rotor

After volume mesh in gambit, results of meshing were checked. There were a total of 959040 elements, from which the number of skewed or inverted elements was zero.

Numerical Method

Numerical Simulations are done on the ANSYS fluent. ANSYS fluent uses finite volume method, using pressure based solver in order to solve the Navier Strokes equations. Within the fluent, in order to process the turbulence effect realizable k-epsilon turbulence model with scaleable wall function as a wall treatment is used. For the meshing purpose, grids are generated in the pre-processing software Gambit. In order to evaluate the performance of the turbine numerically, there are four dimensionless coefficients i.e. pressure coefficients C_A, torque coefficients C_T and turbine efficiency η and flow coefficients ϕ(axial velocity/rotational velocity). These can be calculated from the formulas given below [S Shaban,2017].

$$C_T = \frac{T}{\frac{1}{2}\rho\Omega^2 r_{\text{tip}}^5}$$

$$C_A = \frac{\Delta p}{\frac{1}{2}\rho\Omega^2 r_{\text{tip}}^2}$$

$$\eta = \frac{C_T}{C_A \phi} \frac{1}{\pi\left(1 - \frac{r_h^2}{r_t^2}\right)}$$

The newly designed turbine is analyzed using ANSYS Fluent. In order to analyze this sort of rotating bodies system, Moving Frame Reference system is used. The boundary conditions considered in the benchmark is, inlet velocity, speed of rotation and the outlet pressure, where the cell zone condition taken from the benchmark [S Shaaban,2016].
Turbine rotates with 800 rpm along the y- axis direction. The inlet velocity is varying from 8 to 12 m/s, and the outlet pressure is set as gauge pressure.

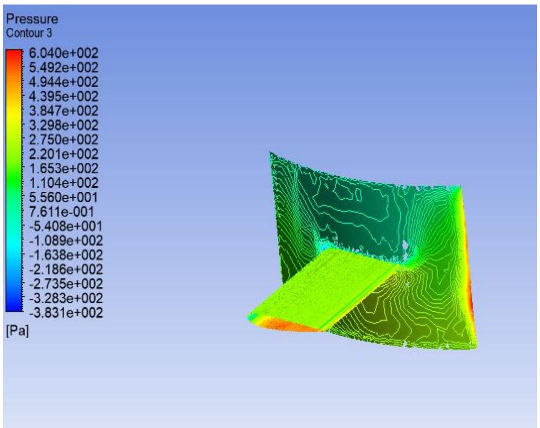

Figure 5 pressure contours at 0.1 Flow Coefficient

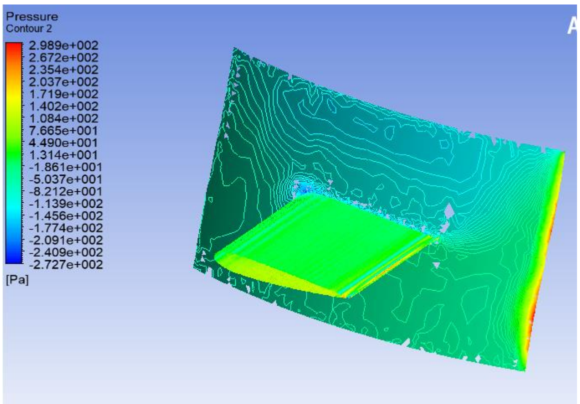

Figure 6 pressure contours at 0.14 Flow Coefficient

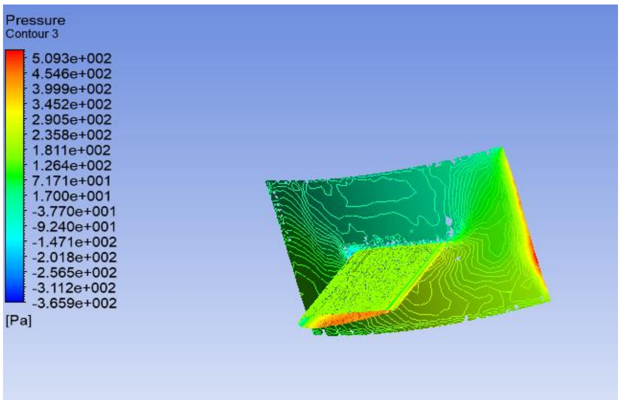

Figure 7 pressure contours at 0.18 Flow Coefficient

Results of the existing NACA 0015 were obtained by copying geometry taken from [S.Shaaban,2017]. and perform same Numerical and meshing scheme and compared with the results obtained from the improved design done in ANSYS fluent. From the analysis we calculated net torque coefficient, force acting on the turbine and the area with the help of fluent software in ANSYS16. The actual performance is estimated from this step considering the flow of air around the airfoil. Simulations of turbine were examined on different inlet velocities varying from 6 to 12 m /s. The turbine performance under steady flow conditions is approximated by the parameters given in the table below.

Table 2 Results at three different values of Flow Coeffcient

Flow Coeff.	Presure Coeff.	Torque Coeff.	Efficiency
0.1	0.218719	0.024812	0.34425257
0.14	0.391796	0.02816	0.513404396
0.18	0.612653	0.032232	0.292277781

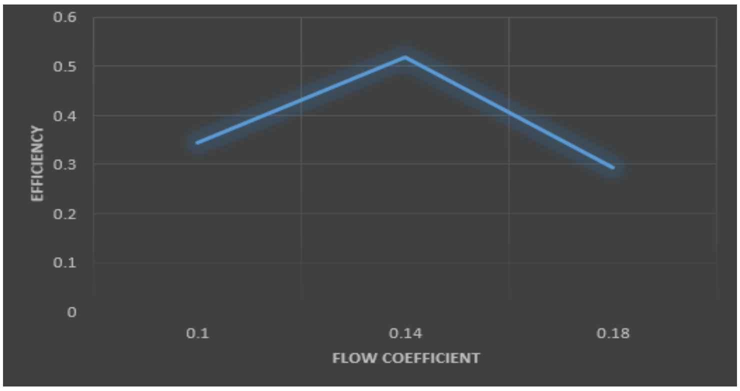

Figure 8 Efficiency Graph

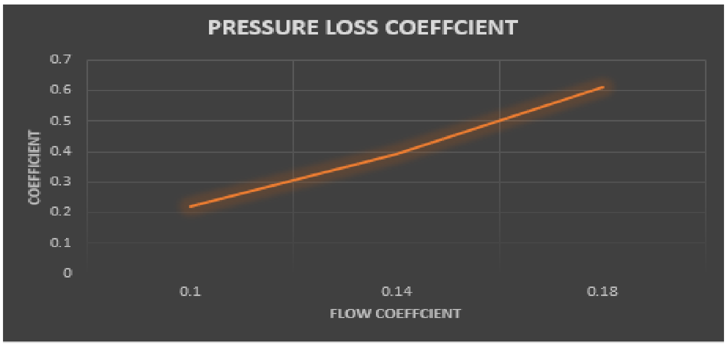

Figure 9 Pressure loss Graph

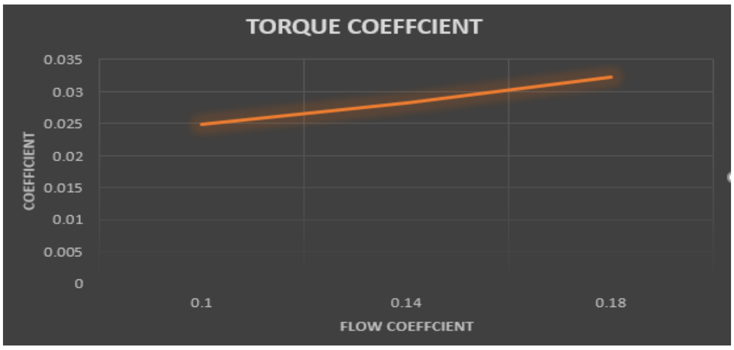

Figure 10 Torque Coefficient Graph

VALIDATION RESULTS

For validation process we have followed the same procedure for NACA 0015 used as for optimized profile. Analysis on NACA 0015 is already done [S shaban 2017]. The purpose of validation is to evaluate the procedure, conducted for the analysis of optimized airfoil.
Results were taken for different values of flow coefficients i.e. 0.1, 0.14 and 0.18. For all these 3 values of flow coefficients corresponding the values of pressure drop and torque coefficients were calculated and so the efficiency from the formulas mentioned above are:

Table 3 Results for Validation

Flow Coeff.	Pressure Coeff.	Torque Coeff.	Efficiency
0.1	0.231645	0.01528	0.499718614
0.14	0.386268	0.023762	0.439397777
0.18	0.588345	0.03171	0.299414146

CONCLUSION

From this it is concluded that the efficiency of the turbine is changed by varying flow coefficient, and control points along the chord length. It is also concluded that the efficiency of the optimized design increases by changing location of maximum thickness close to 0.2c. The overall maximum efficiency using such technique was found to be around 50% at 0.14 flow coefficient. Hence, this is a practical way of extracting energy from waves and is an economical option.

References

Curran, R., & Gato, L. M. C. (1997). The energy conversion performance of several types of Wells turbine designs. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, *211*(2), 133–145. https://doi.org/10.1243/0957650971537051

Dawoud, B., Amer, E., & Gross, D. (2007). Performance analysis of CCHP and CHP systems operating following the thermal and electric load. *International Journal of Energy Research*, *31*(August 2007), 135–147. https://doi.org/10.1002/er

Johnson Sriram Kumar, S. S., Thachuparambil Vivek Joseph John, T. B., & Anil Kumar, G. (2015). Design and Analysis of 3D Blades for Wells Turbine. *IJIRST –International Journal for Innovative Research in Science & Technology*|, *1*(11), 202–207.

Shaaban, S. (2016). Aero-economical optimization of Wells turbine rotor geometry. Energy Conversion and Management Volume 126, 15 October 2016, Pages 20-31 https://doi.org/10.1016/j.enconman.2016.07.068

Shaaban, S. (2017). Wave energy harvesting using a novel turbine rotor geometry. *International Journal of Energy Research*, *41*(4), 540–552. https://doi.org/10.1002/er.3638

Shehata, A. S., Xiao, Q., Saqr, K. M., Day, A., & Shehadeh, M. F. (2015). Performance analysis of wells turbine blades using the entropy generation minimization method. *Renewable Energy*, *86*, 1123–1133. https://doi.org/10.1016/j.renene.2015.09.045