



CFD Based Optimization of Kaplan Turbine Blade Profile

Zahid Masood¹

¹College of Electrical and Mechanical Engineering, Hohai University, Changzhou 213022, China
zahidmasood@gmail.com¹

Received: 23 December 2019, Revised: 01 February, Accepted: 05 February

Abstract— Living in the 21st century and having huge potentials of renewable energy resources, people in this part of the world are still deprived of the basic needs of life, among those clean sources of energy is one. Renewable systems have direct local environment benefits in terms of substituting for existing diesel-based power generation. The efficiency of a turbine is highly influenced by its runner. The focus of the study is to design and optimize the Kaplan turbine runner. First, a theoretical design was performed for determining the main characteristics where it showed an efficiency of 4%. theoretical equations are generalized and simplified and also, they assumed constants of experienced data and hence a theoretical design will only be approximate. This was confirmed as the same theoretical design showed only 57% of efficiency when CFD simulation was run on the model. Then, the theoretically proposed design was further analyzed, inlet/outlet tangential velocities of the blades were analyzed and corrected with CFD Fluent to improve the efficiency of power generation. The original design could be improved to achieve an efficiency of 86%. In the end, a more suitable and appropriate method was suggested to optimize the design.

Keywords— Renewable energy, Hydropower Plants, Hydro Turbines, Design and optimization

I. INTRODUCTION

There is a strong requirement to change fossil fuels into renewable energy in recent years because of the exhausting energy resources and terrible environmental pollution. Reservoirs of conventional energy resources are decreasing rapidly with every passing day. This present energetic situation leads the research for renewable energy sources. Renewable energy resources can be used as alternatives to conventional energy resources [1]. Renewable energy resources are most cheap and pollution-free sources of energy. The extraction of penitential and kinetic energy from water flow is particularly interesting, called hydropower. Hydropower is one of the most popular, oldest, largest, cheap, reliable and efficient energy sources among other Renewable Energy Resources.

Hydropower Plants can be found all over the world rather than Wind or Solar Power Plants. Also, Hydropower systems can achieve efficiency by up to 95%. [2]

Hydro Turbines can be divided into three major types (Kaplan, Francis & Pelton Turbines) and the selection of right and suitable turbine to install in a Hydropower system depends on the head (H) and a coefficient related to specific speed (σ). [3]

Head (H) is the difference between the Headrace water level and the Tailrace water level. Coefficient (σ) mainly depends on the turbine's rotational speed (N), Discharge rate (Q).

$$\sigma = \frac{2N\sqrt{\pi Q}}{(2gH)^{3/4}}$$

II. DESIGN OF TURBINE BLADE

For the purpose of this study a Kaplan Turbine installed inside a run-off the river-based Hydropower Plant is considered with the head (H) of 4.2 m and discharge (Q) of 40 m³s⁻¹.

A. Diameter & Rotational Speed

The rotational speed can be obtained by using the below equation;

$$N = \frac{\sigma(2gH)^{3/4}}{2\sqrt{\pi Q}} = \frac{1.45(2*9.81*4.2)^{3/4}}{2\sqrt{3.1416*40}} = 1.7688s^{-1} = 106.1316 \text{ min}^{-1}$$

For Specific Speed,

$$N_s = \frac{N\sqrt{Q}}{H^{3/4}} = \frac{1.7688\sqrt{40}}{4.2^{3/4}} = 3.813$$

Outer Diameter (D_a) can be expressed as;

$$D_a = \frac{2\delta}{\sqrt{\pi}} \sqrt[4]{\frac{Q^2}{2gH}} = \frac{2*1.3}{\sqrt{3.1416}} \sqrt[4]{\frac{40^2}{2*9.81*4.2}} = 3.0792m$$

Using coefficient related to specific speed (σ), diameter number (δ) as a reference, Diameter Ratio (DN/D_a) can be read from figure 5.

$$\frac{D_N}{D_a} = 0.4$$

$$D_N = 0.4D_a = 0.4 * 3.0792 = 1.2316m$$

Theoretically, blades profile can be obtained using velocity triangles at the leading and trailing edges and also in the middle of the blade. In This case, each edge is divided into 5 different diameters (D_a D_1 D_2 D_3 D_N). Each blade section at different radii will have different velocities, angles and chord shapes as shown in figure 1.

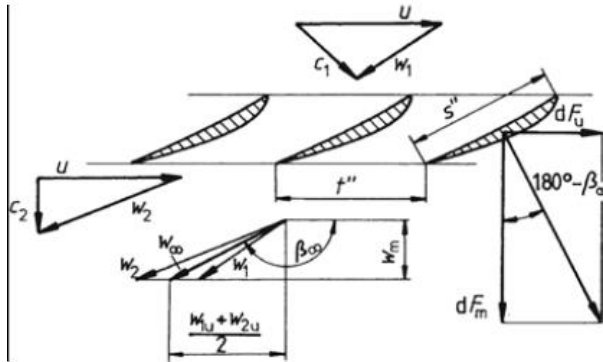


Figure 1: Schematic of relevant velocity triangles[4]

TABLE I. USING THE VELOCITY TRIANGLE APPROACH, THE FOLLOWING VALUES CAN BE OBTAINED IN BELOW TABLE

	D_a	D_1	D_2	D_3	D_N	Units
D	3.0792	2.6173	2.1554	1.6935	1.2316	m
u	17.1106	14.5439	11.9772	9.4105	6.8438	m/s
C_{u1}	3.5397	4.1644	5.0568	6.4361	8.8499	m/s
C_{u2}	1.2762	1.5015	1.8232	2.3205	3.1908	m/s
C_{u∞}	2.4080	2.8329	3.4400	4.3783	6.0203	m/s
W_{u1}	-13.5709	-10.3795	-6.9204	-2.9744	2.0061	m/s
W_{u2}	-15.8344	-13.0425	-10.1540	-7.0900	-3.6530	m/s
W_∞	16.0330	13.3430	10.6664	8.1371	6.4473	m/s
W_m	6.3945	6.3945	6.3945	6.3945	6.3945	m/s
W_{u∞}	-14.7027	-11.7110	-8.5372	-5.0322	-0.8235	m/s
ΔW_u	2.2635	2.6630	3.2336	4.1156	5.6591	m/s
β_∞	156.4949	151.3645	143.1664	128.2016	97.3382	°
β₁	154.7707	148.3642	137.2620	114.9459	72.5824	°
β₂	158.0095	153.8822	147.7994	137.9528	119.7385	°
s''/t''	0.75	0.888	1.025	1.163	1.300	
t''	2.4184	2.0556	1.6928	1.3301	0.9673	m
s''	1.8138	1.8254	1.7352	1.5469	1.2575	m

III. CAD DESIGN AND ANALYSIS

A. CAD Model

Using information available in the above table, it's easy to construct the blade profile. Three spline curves were generated in SolidWorks, then combine these curves with a surface. Thus, it creates a profile of a turbine blade. The CAD model is shown in figure 2.

B. Fluent Analysis

To perform CFD analysis commercial ANSYS Fluent software was used. For the meshing tetrahedral meshing approach was used because for this kind of problem [5], the tetrahedral mesh has proven more suitable as shown in the figure. 3.

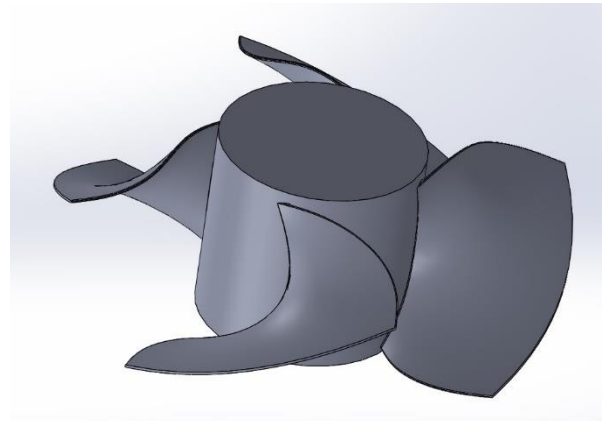


Figure 2: CAD Model

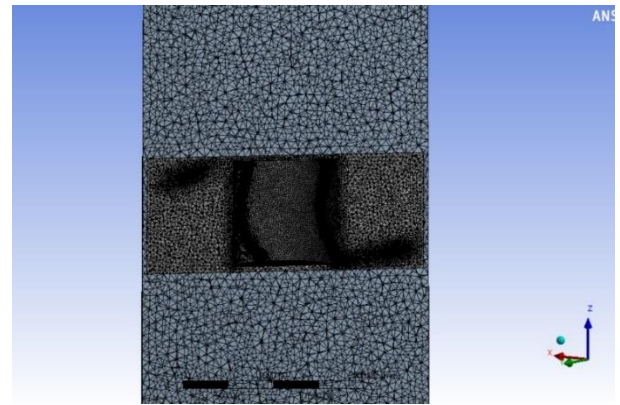


Figure 3: Meshing

IV. OPTIMIZATIONS

Transit state simulations were run using k- ϵ 2-equations to compute the results and get tangential velocities and leading edge and trailing edge of the blade.[6]

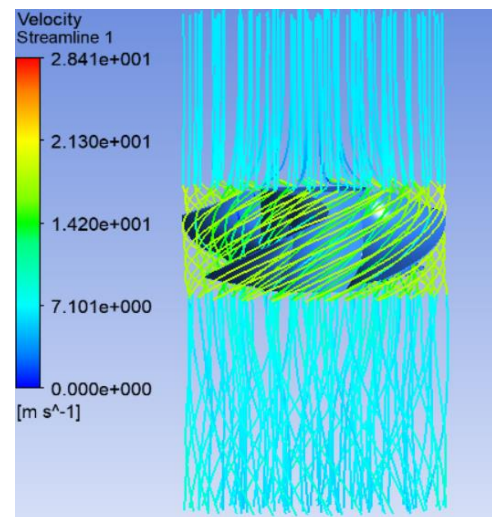


Figure 4: Velocity streamlines

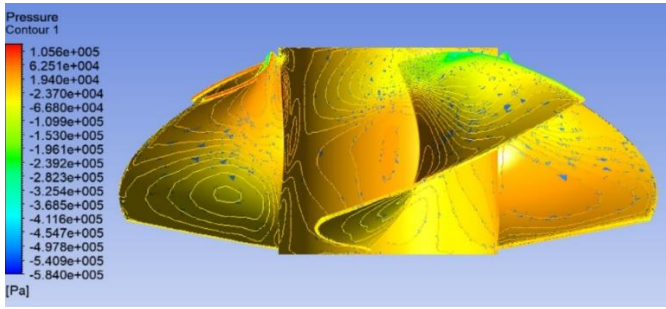


Figure 5: pressure contour

By computing the results from Figures 4 and 5, the efficiency of this model came out to be 57%, which far less than the theoretical value. This can be improved by changing the inlet Velocity, a number of blades or the shape of the blade. [7]

C. Inlet Velocity

First, we try to change the inlet velocity, run CFD analysis using the same conditions used and observe the behavior. Results are given below in Table II and figure. 6

TABLE II.

C_0 (ms^{-1})	5.50	6.00	6.395	7.00	7.50	8.00	8.50
Δw_u (ms^{-1})	1.70 63	1.7991	1.740	1.8360	1.8244	1.8072	1.7823
η (%)	52.7 7	55.64	54	56.78	56.42	55.89	55.12

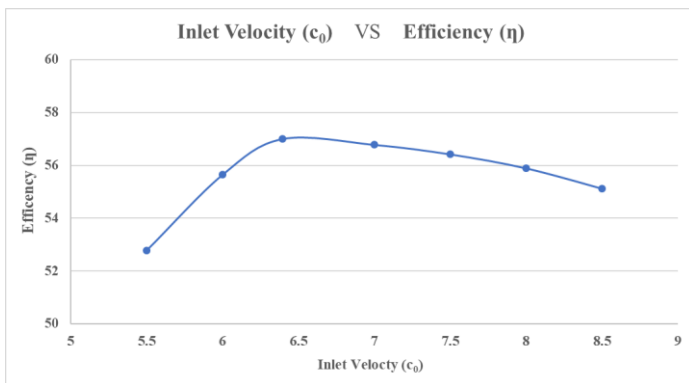


Figure 6: Inlet velocity's affect on efficiency

From the graph, it can be seen that increasing the velocity reduced efficiency, which means increasing the inlet velocity is suitable.

D. Number of Blades

Another way to improve efficiency is to increase the number of blades. The results can be seen in the following observations in Table III and figure 7.

TABLE III.

Z''	3	4	5	6	7
Δw_u (ms^{-1})	1.3788	1.74	2.1286	2.2140	2.1594
η (%)	42.64	54	65.83	68.47	66.78

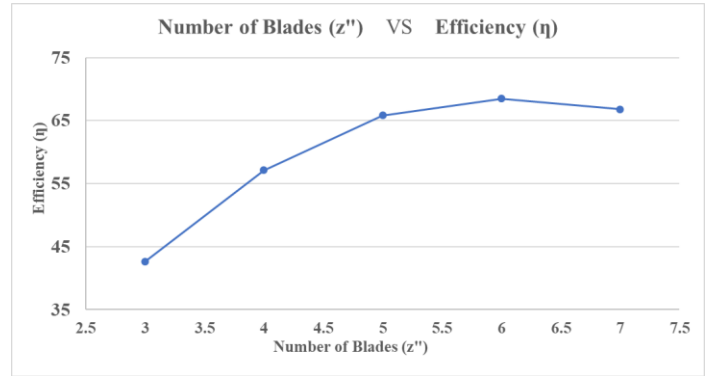


Figure 7: Effect of Number of blades on Efficiency

After analyzing the results, it can be seen that increasing the number of blades can increase efficiency but for this type of turbine best efficiency should be achieved using 4 number of blades. [8]

E. Blade Edge Angles

To change the shape of the blade best option is to change the β_1 and β_2 . This will change the behavior of fluid over the blade and will increase the drag and lift coefficient, resulting in increased efficiency.

Each blade edge is divided into 5 parts, and there are 2 edges (inlet & Outlet Edge), so there are millions of possible combinations of angles to construct models. This method is long and time-consuming. Based on intentions, different angle combinations were used to construct different design models and run CFD simulations using ANSYS Fluent using the same conditions used and try to find the maximum efficiency. The results are given below in Table IV.

TABLE IV.

Δw_u	1.740	1.827	2.317	2.449	2.369	2.700	2.712	2.714
η	54%	56.77%	71.91	76.03%	73.58%	83.78%	84.13%	83.21%

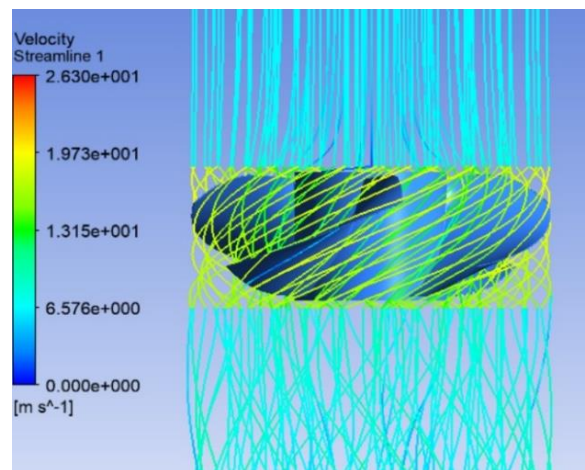


Figure 8: Optimized design velocity streamline

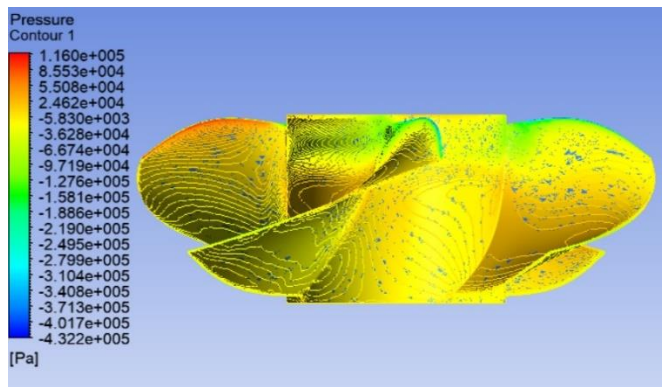


Figure 9: Optimized design pressure contour

CONCLUSION AND FUTURE SCOPE

Theoretically, our model should have 94% but our initial analysis shows the 57% efficiency. This can happen for numerous reasons. Then we try to maximize the efficiency by hanging variables depending on our intuition and achieve almost 84%. This approach is not much accurate and also needs a lot of computational time because of there a million combinations of our changing variables. The more accurate technique can be applied like Surrogate-based model optimization, which can give more accurate and better results with less computational time. The next paper will be focused on using this dimension reduction and surrogate model-based optimizations to get more accurate results.

REFERENCES

- [1] "Hydroelectric Power," *Invest. Energy*, pp. 145–152, 2012, doi: 10.1002/9781119204442.ch16.
- [2] P. Breeze, "Hydropower Turbines," in *Hydropower*, Elsevier, 2018, pp. 35–46.
- [3] "Kaplan Turbine - its Components, Working and Application." [Online]. Available: <https://theconstructor.org/practical-guide/kaplan-turbine-component-working/2904/>. [Accessed: 20-Jan-2020].
- [4] S. L. Dixon, *Fluid Mechanics and Thermodynamics of Turbomachinery*, 5th ed. Elsevier, 2005.
- [5] "Engineering Simulation & 3D Design Software | ANSYS." [Online]. Available: <https://www.ansys.com/>. [Accessed: 08-Jan-2020].
- [6] A. B. Janjua, M. S. Khalil, and M. Saeed, "Blade Profile Optimization of Kaplan Turbine Using CFD Analysis," *Mehran Univ. Res. J. Eng. Technol.*, vol. 32, no. 4, pp. 559–574, 2013.
- [7] H. Nagpurwala, "Design of Hydraulic Turbines."
- [8] K. Menny, *Hydraulische und thermische Kraft-und Arbeitsmaschinen*. Germany: Verlag, Ronnenberg, 2006.