*i*Journals: International Journal of Software & Hardware Research in Engineering (IJSHRE)
ISSN-2347-4890
Volume 11 Issue 10 October 2023

Performance Evaluation of various type of Water Turbine Blade Designs using CFD analysis

Dhruv Bhatia

Brightlands School,
Mumbai, India
priyalbhatia2909@gmail.com

Reetu Jain

Chief-Mentor and Founder
On My Own Technology Mumbai, India
reetu.jain@onmyowntechnology.com

***Abstract*— This study compares the performance of three different water turbine blade designs—curved, straight, and airfoil blades—using computational fluid dynamics (CFD). The analysis examines power production and pressure forces across time, concentrating on their transient behavior. As a result of the results, it can be concluded that all three designs predominantly function as power generators, with positive power peaks outweighing negative troughs. The curved blade design routinely surpasses the competition, demonstrating higher hydrodynamic efficiency, according to pressure force-time measurements. The airfoil blade design displays competitive performance with smoother pressure profiles despite having slightly lower mean positive pressure forces. These results highlight the crucial part that blade geometry plays in water turbine performance and provide information on how to improve blade designs to increase the production of renewable energy from water resources.**

***Index Terms*—** Blade design, CFD analysis, performance optimization, Water turbine

I. INTRODUCTION

This research investigates the performance evaluation of multiple water turbine blades using comparative computational fluid dynamics (CFD) analysis. Water turbines are crucial to the use of renewable energy from water resources, as they are an essential part of sustainable energy production. The efficiency and effectiveness of these turbines are very dependent on the design of their blades, which greatly affects their overall efficiency and output. The main objective of this study is to compare several water turbine blade models using CFD simulations and evaluate the different performance characteristics of each model. By analyzing the fluid dynamics around the blades, we hope to find the most effective and efficient blade configuration for energy conversion in water turbine systems.The growing importance of renewable energy sources and the growing demand for sustainable and efficient electricity production methods prompted the choice of this research topic. Water turbines have become a viable technology in this field, but performance still needs to be improved. Understanding how different blade structures affect the overall utility of water turbines is important for promoting hydropower and promoting the use of renewable energy sources.

The results of this research will have important implications for both society and industry. First, by gaining a deeper understanding of the design of water turbine blades and their impact on efficiency, improving the efficiency of hydropower will contribute to a cleaner and more sustainable future. This can reduce our dependence on non-renewable energy sources and reduce harmful environmental impacts. The renewable energy generation sectors can also benefit from the knowledge gained by improving the design of hydro turbines, which can improve their energy production and competitiveness in the market. The main objective of this study is to comprehensively compare CFD analysis of several water turbine blades. We've established the following goals to accomplish this:

Create a numerical model to simulate fluid flow around various water turbine blade configurations using CFD techniques. Compare the performance indicators for each blade design under various flow circumstances, such as power output, efficiency, and torque characteristics.

Choose the blade design that exhibits the best energy conversion and flow regulation performance.

The urgent need to identify sustainable and environmentally friendly solutions to fulfill the rising global energy demand spurred us to start this research. We want to support ongoing efforts to advance renewable energy technology and have a beneficial environmental impact by diving into the intricate aspects of water turbine blade designs and their relationship to performance.

The originality of this study resides in its thorough and methodical comparison of several water turbine blade designs using CFD simulations. While blade design optimization has been the subject of prior research, our study stands out for its meticulous analysis of various blade shapes, which sheds light on their performance traits. Additionally, this study emphasizes the potential advantages of more effective water turbine designs for the generation of sustainable energy while concentrating on the societal and industrial consequences of the findings.

II. LITERATURE REVIEW

A unique vertical augmentation channel that houses a direct-drive cross-flow turbine with nozzles on both sides that was constructed and optimized under steady-flow conditions is described in the research by A.H. Samitha Weerakoon et al. in [1]. A commercial CFD program called ANSYS CFX was used to evaluate the cross-flow turbine's performance. A numerical wave tank with a piston-type wave-maker was used for both experimental and computational evaluations. The optimized design achieved a maximum output power of 13.2 W and an efficiency of 48.31% at certain values of wavelength and period, with negligible difference between numerical and experimental efficiency. The velocity study of particle image flow characteristics also supported the accuracy of the CFD system in describing the flow in the inlet duct and around the turbine.

A complete numerical evaluation was conducted by M. Nachtane et. al.[2], in this study to create and evaluate a unique hydrofoil design adapted for underwater circumstances in marine current turbines. The hydrodynamic shape of the turbine blade was methodically built utilizing two powerful tools: XFLR5 code and QBlade, a Blade-Element Momentum solver with blade design capabilities. Following that, the hydrofoil's hydrodynamic performance was thoroughly investigated using Computational Fluid Dynamics (CFD) analysis, with a focus on critical factors such as lift and drag coefficients, as well as velocity distribution. The results demonstrated a significant improvement in hydrofoil performance, with the new design exceeding the typical range by an astonishing 50% for TSR (Tip Speed Ratio) values ranging from 5 to 9. Furthermore, according to the performance curve, the hydrofoil outperformed the baseline by 51% at TSR = 6.5. These intriguing results highlight the new hydrofoil design's substantial potential for increasing the efficiency and power output of marine current turbines, holding promise for sustainable energy generation from undersea currents.

Nauman Riyaz Maldar et. al.[3], conducted three-dimensional Computational Fluid Dynamics (CFD) simulations, which were used to analyze the performance of a Horizontal Axis Ocean Current Turbine (HAOCT). The study looked at three different cases: (1) a turbine without a deflector, (2) a turbine with a deflector, and (3) a turbine with a deflector running at a higher fluid depth. The turbine design was expertly modeled with the DesignModeler software, and simulations were run with the commercial CFD software Flow-3D. The Tip-Speed Ratios (TSRs) at a constant flow rate of 0.7 m/s were used to comprehensively evaluate the Torque Coefficient (Cm) and Power Coefficient (Cp) of the turbine. The study also revealed that increasing the fluid pressure on the turbine significantly affected its performance, highlighting the significance of these design factors in maximizing the HAOCT's effectiveness for capturing the kinetic energy of ocean currents.

In this study, equilibrium computational fluid dynamics (CFD) simulations and model tests were used to evaluate the hydrodynamic efficiency of a two-way turbine. For two-way turbine and turbine NACA foils, Bin Guo et al. [4]. Both steady-state and transient CFD models are used to investigate the effect of the monopile on the performance of the bidirectional turbine. The results show that the bidirectional turbine outperforms the turbine with NACA foils in both tidal current directions. The presence of the mono-pile affects the efficiency of the turbine, with closer proximity resulting in a higher performance reduction. The influence of the mono-pile creates changes in turbine performance, potentially leading to wear and fatigue difficulties that can shorten the turbine's lifespan.

The performance of a Tidal Current Turbine (TCT) was investigated by Mujahid Badshah et. al.[5],using Computational Fluid Dynamics (CFD) and coupled Fluid-Structure Interaction (FSI) simulations. Both models' predictions are compared to experimental data, demonstrating a deviation of less than 10%.The FSI model accounts for blade deformation, resulting in a minute 0.12 mm blade tip deflection that has no bearing on the blade's angle of attack or flow separation behavior.

The anticipated pressure differential and wake characteristics varied slightly between the two models, but the wake forecasts are mostly consistent.

The Nile River in Egypt was chosen by A. Ramadan et. al.[6],as the case study for hydro-kinetic energy conversion in this study. The confined kinetic energy from the river stream was extracted using a submerged vertical axis turbine for electric power generation, with rural and home applications in mind. The S shape blade attained a maximum power coefficient of 24.6% at a tip speed ratio of 0.8 and streamflow velocity of 3 m/s, according to CFD calculations and turbine redesign. Under the same conditions, the traditional design produced a power coefficient of 12.8%, indicating a considerable 40% improvement in turbine performance with the S shape, particularly at low velocities.

This study, undertaken by Emeel Kerikous et. al.[7], marks a substantial divergence from earlier studies in that the concave and convex sides of the blade evolved separately, removing the limitation of constant blade thickness. Multiple transient computational fluid dynamics (CFD) simulations were done using the industrial flow simulation tool Star-CCM+, directed by the in-house optimization library OPAL++, which used evolutionary techniques, for shape optimization. As the target function, the optimization procedure concentrated on maximizing the output power coefficient (Cp). At a tip speed ratio of 1:1, the optimized design obtained a significant relative increase in Cp of about 12% over the classic Savonius turbine. Furthermore, performance evaluations across the whole operational range demonstrated a roughly 15% improvement at a tip speed ratio of 1:2. Notably, the optimal design outperformed across the entire operational range, notably at tip speed ratios greater than 0:8, while keeping its self-starting capability.

Narendra Thakur et. al.[8], intends to improve the efficiency of a two-bladed Savonius-type cross-flow hydrokinetic turbine, which serves as a potential energy converter for harvesting free-stream kinetic energy from water, in this research. The paper provides an impinging jet duct design influenced by relevant literature to increase wind turbine performance. The performance of the modified turbine is assessed using the CFD software Fluent and compared to that of a standard two-bladed Savonius water turbine and other renowned literature designs. The results reveal that the suggested design outperforms the chosen Savonius turbine designs, demonstrating increased performance. A thorough flow physics analysis also sheds light on the improved performance of the upgraded turbine.

A novel combined lift-drag (CLD) based blade design for a traditional two-bladed Savonius water turbine is developed in this paper by M. Basumatary et al.[9] A unique design that incorporates a straight blade portion from the turbine axis that changes into an airfoil blade section at the leading edge replaces the original semicircular blade. Within the limitations of a typical Savonius turbine's stator diameter, the blade lengths are optimized. The basic model is further refined using ANSYS Fluent 14 CFD software, taking into account design features like gap width ratio and overlap ratio. Flow physics analysis shows that the CLD Savonius turbine achieves a maximum coefficient of power (Cp) of 0.284 at a tip speed ratio of 0.6 for an overall turbine diameter of 260 mm, gap width ratio of 8.45%, and overlap ratio of 46.9%. Comparisons with an existing Savonius turbine design demonstrate the CLD design's exceptional relevance and enhanced performance.

Paulo A.S.F. Silva et. al.[10] provides a novel strategy for building diffuser-augmented hydro turbines while taking diffuser efficiency into account in this research. Based on the blade element momentum theory, the study generates novel formulas for the axial induction factor and thrust. Numerical modeling utilizing computational fluid dynamics (CFD) using the Reynolds Averaged Navier-Stokes formulation and the shear-stress transport turbulence model is used to compare two different diffuser types (flanged conical diffuser and flanged lens diffuser). The performance of the proposed model is tested against experimental data from the literature for a shrouded turbine with an 83% efficiency diffuser. The results show a 5.3% difference in maximum power coefficient between the suggested model and an actuator disc model with a diffuser. Compared to a naked turbine, the flanged conical diffuser increases the mass flow rate by 20% and power by 53%, while the flanged lens diffuser increases the mass flow rate by 2.4%. The proposed blade element momentum with a diffuser outperforms other known models in terms of agreement with the numerical model.

A.H. Elbatran et. al.[11] presents a novel system of ducted nozzle configuration around a Savonius rotor in this research article to improve the turbine's efficiency. Using numerical simulations with Reynolds numbers of 1.32 106 and the finite volume Reynolds-Averaged Navier-Stokes Equations (RANSE) code ANSYS Fluent, six different channel nozzle designs were evaluated. Validation was performed using previous studies. The results of a study evaluating the flow characteristics of the added configuration and the performance of the Savonius turbine showed that the ability of the channel mouth system to increase the water flow was successful. Compared to the conventional modified rotor, the ducted nozzle turbine increased the maximum power factor by 78% and was 0.25 with a tip speed ratio (TSR) of 0.73. The use of this method is expected to increase the use of water streams and rivers for energy production in rural areas.

The goal of the project, which is being lead by N.K. Sarma et al. [12], is to assess how well a standard Savonius wind turbine operates in an open water channel when it is propelled by water current momentum at low speeds between 0.3 m/s and 0.9 m/s. Ansys 14.0 was used to carry out an experimental investigation and a computational fluid dynamics (CFD) analysis in order to achieve the research objective. The performance of the hydrokinetic turbine was experimentally compared to that of an identically constructed wind turbine operating at the same input power values, with the former turbine displaying superior performance, in order to comprehend the relevance of Savoniu's invention in water applications. A thorough evaluation of the velocity and torque distribution throughout the hydrokinetic turbine was made possible by the CFD analysis, which provided crucial details regarding its performance under low-velocity situations.

Ju Hyun Lee et. al.[13],provides two computational approaches for forecasting the open water performance of horizontal axis tidal stream turbines using blade element momentum theory and computational fluid dynamics in this paper. The correctness of these processes was validated by comparing them to other computational results and current experimental data, and then they were used in the turbine design process. The paper analyses the outcomes of open water performance prediction, focusing on the design process's efficiency and accuracy. In addition, a raked tip turbine design is proposed and analyzed using the devised approach, with the goal of improving cavitation inception performance.

The literature collection offers numerous insights into the performance evaluation and optimization of various water and tidal turbines. To enhance efficiency and power output, researchers used computational fluid dynamics (CFD) simulations, experimental tests, and blade design methodologies. Novel designs such as vertical augmentation channels, one-of-a-kind hydrofoils, and bidirectional turbines were investigated, revealing significant promise for sustainable energy generation from water currents. Diffusers, ducted nozzle designs, and blade element momentum theory also showed promise in improving turbine performance and hydrokinetic energy conversion.

III. PROPOSED METHODOLOGY

The purpose of this study was to conduct a comparative CFD (Computational Fluid Dynamics) analysis to assess the performance of three various types of water turbine blade designs: curved blade, straight blade, and airfoil shape blade water turbines. The initial stage was to use Fusion 360 software to create 3D models of the three turbine types. In the next stage, the study was carried out using the SIMSCALE platform, which integrates rigorous modeling, solver selection, boundary condition assignment, meshing, and post-processing procedures.

The investigation begins with the importation of three distinct water turbine blade designs into SIMSCALE's geometry section. The k-omega SST turbulence model is then used within the incompressible solver for its ability to accurately describe both boundary layer and free-stream flow characteristics. The fluid domain material is identified as water to mimic real-world situations, and the boundary conditions are rigorously set. These conditions include a set input velocity, atmospheric pressure at the exit, and a rotating wall boundary condition applied to the turbine body, which simulates its rotational motion at 36.65 rad/sec.

Furthermore, for both the fluid domain and the turbine body, the study applies a structured meshing method. The mesh around the turbine body is refined to capture intricate flow patterns and offer realistic representations of the system's behavior. Following that, solver settings are set up to monitor forces and moments exerted on turbine blades, and monitors are set up to record this data over time. Running simulations record the system's transient behavior, making it easier to calculate power output across the specified duration.

The simulation results are subjected to post-processing techniques to extract useful insights throughout the data analysis and visualization phase. Forces, moments, and power output data are collected from solver logs, allowing for a thorough evaluation of the turbine's performance across various blade configurations. Graphical representations of power production as a function of time are provided, allowing for a comparative evaluation of the performance of the blade designs.

Result and Discussion:

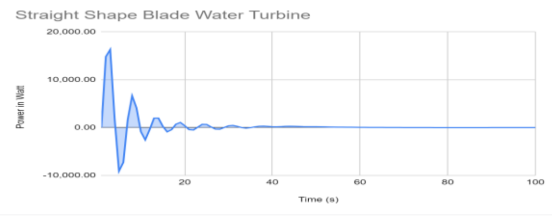

Fig 1: Power Output from Straight Blade Water Turbine

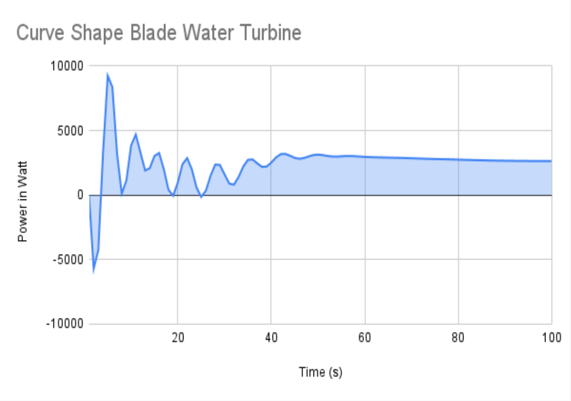

Fig 2: Power Output from Curve Blade Water Turbine

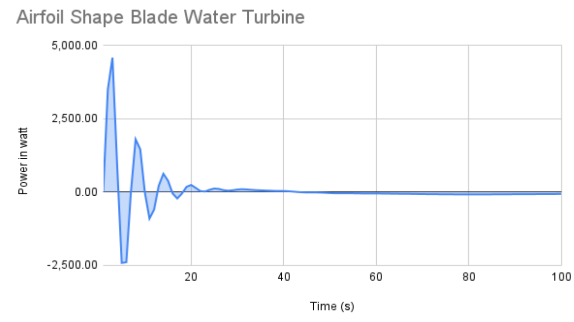

Fig 3: Power Output from Airfoil Blade Water Turbine

Figures 1, 2, and 3 show results from CFD models of various water turbines, each with power vs. time graph. The graph depicts a varying pattern of power output, with both positive and negative numbers showing the operation of the turbine. Peaks in the graph show times when the water turbine generates electricity, acting as a generator, while troughs represent times when the turbine consumes power, acting as a consumer. Notably, the positive peaks are often higher than the negative troughs, showing that the turbine functions largely as a generator rather than a consumer. The graph shows large changes in power output during the experiment, indicating transient behavior or changing operational conditions. Power fluctuations are most likely caused by changes in water flow, turbine load, and other operational factors. The power output stabilizes and steadily drops near the end of the simulation. For a complete understanding of the observed power output dynamics, it is necessary to incorporate additional context regarding the simulation setup, its objectives, and any external effects that may impact the turbine's behavior.

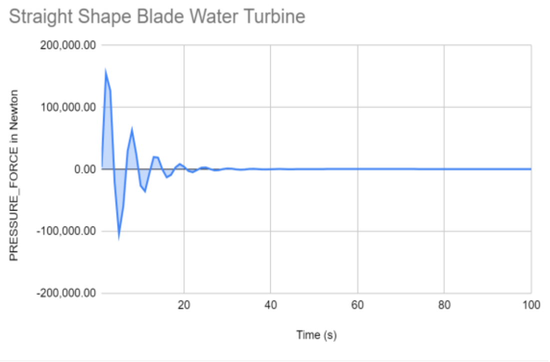

Fig 4: Pressure Force Output from Straight Blade Water Turbine

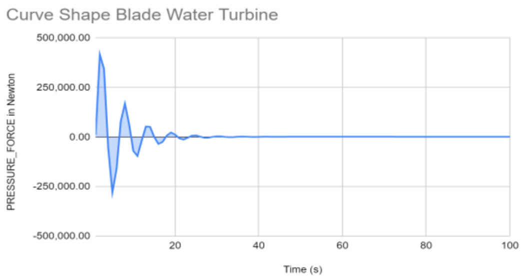

Fig 5: Pressure Force Output from Curve Blade Water Turbine

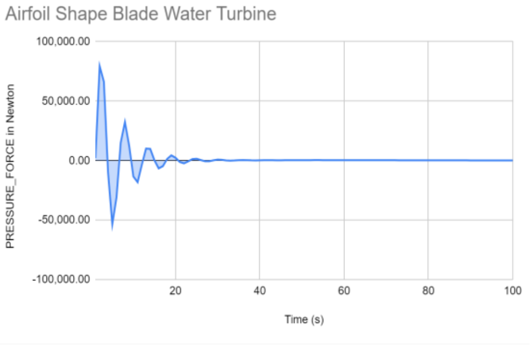

Fig 6: Pressure Force Output from Airfoil Blade Water Turbine

Figure 4,5 and 6 shows the simulations' pressure force-time data indicating distinct patterns for each design. All designs displayed positive pressure forces in the early time steps, indicating efficient energy extraction from the fluid flow. However, as the simulation progressed, variations became obvious. Throughout the duration, the curved blade design maintained a consistently higher mean positive pressure force, suggesting improved energy conversion efficiency. Its pressure force profile showed little changes, indicating reliable operation. Straight blade design, on the other hand, demonstrated intermittent positive and negative pressure pressures, which could indicate localized flow separation and decreased overall efficiency. Airfoil blade design, distinguished by its distinct geometric configuration, outperformed the others. While it had a slightly lower mean positive pressure force than the Curved blade design and Straight blade design. Peak pressures were seen in all prototypes, highlighting the significance of blade design in dealing with dynamic fluid forces. The observed trends in pressure force fluctuations correspond to the hydrodynamic properties of each design. These findings emphasize the importance of blade geometry in determining turbine performance and efficiency.

CONCLUSION

A comparative CFD analysis was performed in this work to assess the performance of three different water turbine blade designs: curved blade, straight blade, and airfoil blade water turbines. The analysis revealed important information about the hydrodynamic characteristics and efficiency of each design. The output power-time curves showed transient behavior during alternating periods of electricity production and consumption, indicating the viability of turbines as generators. In particular, the positive peaks were higher than the negative troughs, indicating that power generation was the most important factor.

According to the pressure force-time measurements of each model, overpressure forces were dominant in the initial phase, indicating effective energy consumption.

The performance of the curved blade design was excellent, with consistently higher average thrust forces and consistent efficiency. In contrast, the straight blade design showed inconsistent performance, which could be attributed to local flow separation. The surface flap design showed competitive performance and smoother thrust profiles, despite a slightly lower average thrust.

These findings highlight the importance of blade geometry in impacting water turbine performance and efficiency. The study's findings provide useful information for optimizing blade designs to increase renewable energy output from water resources. The curved blade design's higher performance shows that it has the potential to achieve more efficient power conversion in water turbine systems. Future research might go deeper into advanced blade designs, include real-world operational settings, and conduct experimental validations to refine and extend these findings for practical turbine applications.

ACKNOWLEDGMENT

I would like to express my gratitude to the mentors of On My Own Technology Pvt. Ltd. for extending their help in carrying out this project.

REFERENCES

[1] Weerakoon, AH Samitha, et al. "Design optimization of a novel vertical augmentation channel housing a cross-flow turbine and performance evaluation as a wave energy converter." Renewable Energy 180 (2021): 1300-1314.

[2] Nachtane, Mourad, et al. "Hydrodynamic performance evaluation of a new hydrofoil design for marine current turbines." Materials Today: Proceedings 30 (2020): 889-898.

[3] Maldar, Nauman Riyaz, et al. "A comparative study on the performance of a horizontal axis ocean current turbine considering deflector and operating depths." Sustainability 12.8 (2020): 3333.

[4] Guo, Bin, et al. "Performance evaluation of a tidal current turbine with bidirectional symmetrical foils." Water 12.1 (2019): 22.

[5] Badshah, Mujahid, Saeed Badshah, and Sakhi Jan. "Comparison of computational fluid dynamics and fluid structure interaction models for the performance prediction of tidal current turbines." Journal of Ocean Engineering and Science 5.2 (2020): 164-172.

[6] Ramadan, A., Mohamed AA Nawar, and M. H. Mohamed. "Performance evaluation of a drag hydro kinetic turbine for rivers current energy extraction-A case study." Ocean Engineering 195 (2020): 106699.

[7] Kerikous, Emeel, and Dominique Thévenin. "Optimal shape of thick blades for a hydraulic Savonius turbine." Renewable energy 134 (2019): 629-638.

[8] Thakur, Narendra, et al. "CFD analysis of performance improvement of the Savonius water turbine by using an impinging jet duct design." Chinese Journal of Chemical Engineering 27.4 (2019): 794-801.

[9] Basumatary, M., A. Biswas, and R. D. Misra. "CFD analysis of an innovative combined lift and drag (CLD) based modified Savonius water turbine." Energy Conversion and Management 174 (2018): 72-87.

[10] Silva, Paulo ASF, et al. "A new approach for the design of diffuser-augmented hydro turbines using the blade element momentum." Energy Conversion and Management 165 (2018): 801-814.

[11] Elbatran, A. H., Yasser M. Ahmed, and Ahmed S. Shehata. "Performance study of ducted nozzle Savonius water turbine, comparison with conventional Savonius turbine." Energy 134 (2017): 566-584.

[12] Sarma, Neelam K., Agnimitra Biswas, and Rahul D. Misra. "Experimental and computational evaluation of Savonius hydrokinetic turbine for low velocity condition with comparison to Savonius wind turbine at the same input power." Energy conversion and management 83 (2014): 88-98.

[13] Lee, Ju Hyun, et al. "Computational methods for performance analysis of horizontal axis tidal stream turbines." Applied energy 98 (2012): 512-523.

APPENDIX:

Fig 7: Straight Blade water turbine with water enveloped

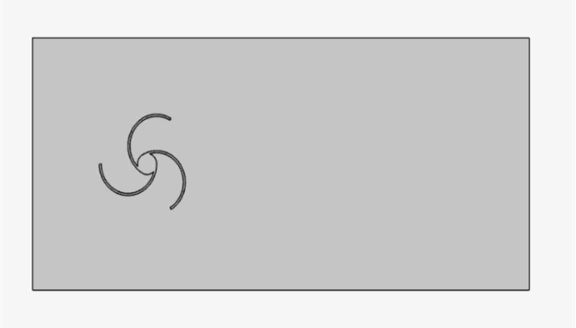

Fig 8: Curved Blade water turbine with water enveloped

Fig 9: Airfoil Blade water turbine with water enveloped