

DESIGN OF A NEW FRANCIS TURBINE RUNNER FOR A REHABILITATION PROJECT IN TURKEY

Elikalfa, A.^{*&}, Sepetci, G.^{*&}, Dr. Celebioglu, K.[&], Assoc.Prof. Aradag S.^{*&}

^{*}Department of Mechanical Engineering

[&]ETU Hydro Energy Research Laboratory

TOBB University of Economics and Technology, Ankara, Turkey

aelikalfa@etu.edu.tr

Abstract: This study presents the rehabilitation project for the Francis turbine runner of Kepez 1 Hydroelectric Power Plant (HEPP) in Antalya, Turkey. The aim of rehabilitation is to increase the power and efficiency of the plant and solve the existing problems such as cavitation. Initially, the head and flow rate of the turbine are estimated. Within the scope of this study, the design parameters which are necessary to obtain better cavitation characteristics and higher power and efficiency, are determined and computational fluid dynamics (CFD) is utilized to design a new runner geometry for better performance. The results of the CFD simulations are compared to the simulation results of the existing runner geometry which was obtained from the plant and scanned using laser scanning. The results show that cavitation characteristics and the efficiency, as well as the power of the HEPP are improved.

Keywords: COMPUTATIONAL FLUID DYNAMICS (CFD), FRANCIS TURBINE, REHABILITATION

1. Introduction

Francis turbines are one of the most common types of hydraulic turbines used in hydroelectric power plants. [1, 2, 3, 4]. Kepez 1 HEPP in Antalya has an existing Francis turbine and this turbine is redesigned and will be implemented on site as a part of a rehabilitation project.

Rehabilitation is commonly defined as increasing the efficiency of the power plant and the energy generation. The latter in this manner is the most economical way of increasing the capacity of the system [5]. The problem about the Kepez 1 HEPP originates from its having been built with the technology of 55 years ago and hence, its being old today. Therefore, the techniques used to design the runner decades ago are very old and of course, state of the art has improved incredibly in time.

There are two choices for rehabilitation projects. First of these is resetting the turbine to the original state and the second is upgrading the turbine components with new components optimized using state of the art technologies. The second choice that is used in this study, is a better solution which can increase the turbine capacity and efficiency and prevent cavitation erosion [6,7,8]. Guide vanes and runner can be replaced easily, but the modification of other components such as spiral case, stay vanes or draft tube is rare [8]. The purpose of runner rehabilitation is to enhance the operating range of the runner, as well as to improve the cavitation limits, power and efficiency.

The flow in the runner is three-dimensional and rotational; it can be computed only by using numerical methods because of its complicated nature [9]. Computational Fluid Dynamics (CFD) is the most developed flow analysis technique today. Flow characteristics of the main components of hydro turbines, their efficiency, tendency for cavitation, rotor-stator interactions and working performance can be examined with CFD [5].

2. Methodology

Numerical studies for Kepez 1 HEPP are performed by using ANSYS-CFX [10] software. The analyses are performed for three-dimensional and steady-state flow. For blade profile, BladeGen [11] software is used and automatic mesh generation is applied by using Turbo Grid [12].

Blade angle and thickness of the runner, as well as the blade design are changed in the new design. Maximum efficiency must be obtained in the middle of the operation range of the turbine. However, the most efficient area is outside of the operating range for the existing turbine. Therefore, it is planned that the efficiency area must be extended as much as possible and it must intersect with the range of operation of the turbine. Friction losses can be reduced and flow separation can be prevented with a smoother blade thickness distribution. The length of the hub can also be shortened for ease of manufacturing. The leading and trailing edges also need to be thickened for the runner to last longer. Fig. 1 shows the thickness distribution for the existing and newly designed turbine runner blades. Fig. 2 shows the comparison of meridional profiles for the two runners.

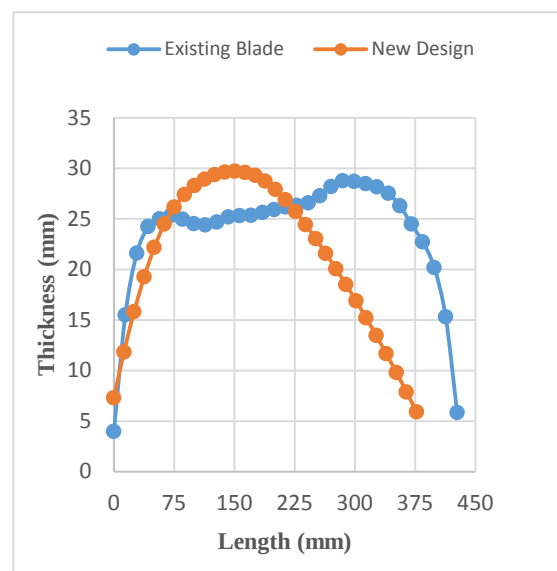


Fig. 1 Thickness distribution of existing and new blade

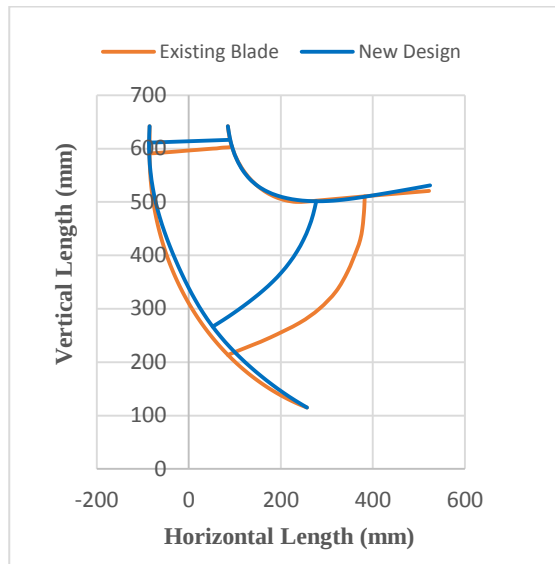


Fig. 2 Comparison of meridional profile of existing and new blades

The pressure gradients reverse their direction on the leading edge of the blade resulting in loss of performance. In order to prevent this situation, X-blade design is preferred. Negative pressure areas in the spans near the shroud are prevented in the new design. This decreases the Thoma numbers and provides a cavitation-free design. The angles in the leading and trailing edges of the blade are also changed to improve the pressure distribution on the blade and to obtain a uniform velocity distribution (meridional and radial velocity).

3. Results & Discussion

The design is first performed for a single runner blade and then full turbine analyses are carried out by inserting spiral case, guide vane and draft tube to the runner.

Analyses are performed for optimum design point (160m head, 6.1 m³/s flow rate and 16.5° flow angle) and power, efficiency, cavitation limit for every modification are examined. A hill chart study is also performed to examine the runner performance for various guide vane openings and flow rates. The boundary condition at the runner inlet is set as total pressure and at the outlet as mass flow rate for single blade analyses. k- ϵ turbulence model is used for all the analyses. Boundary conditions are set as total pressure at inlet and atmospheric pressure at outlet for full turbine computations.

Full turbine analysis is performed for the optimum point of the designed runner and its results are compared with the existing turbine results. Table 1 shows the comparison.

Table 1: General performance values of the existing turbine and the new design (full turbine analysis)

	Power (kW)	Total Efficiency (%)	Head (in-out) (m)
Existing Turbine	8012.4	82.2	151.2
New Design	9022.9	94.1	156.5

The hill-chart of the designed blade, showing flow rate versus head, is given in Fig. 3. As it is seen, the runner has a large operational range with high efficiency.

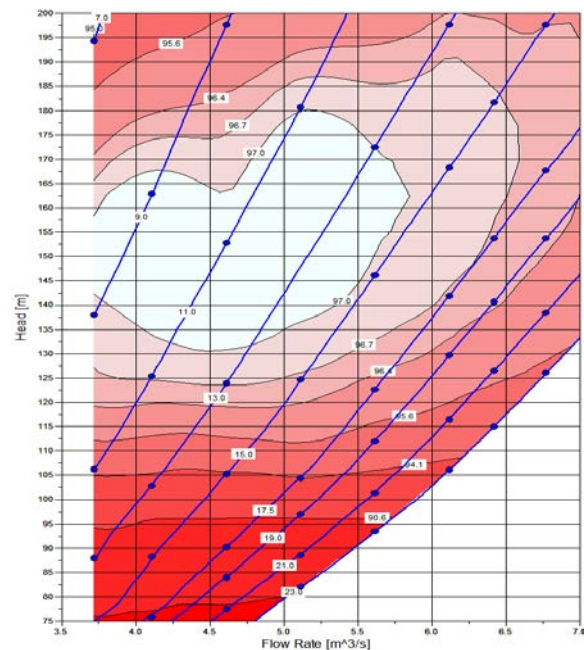


Fig. 3 Hill chart of the new blade

Pressure contours and velocity vectors on the blade are shown in Fig. 4-7. There is no separation or swirl in the flow field and the velocity vectors follow the blade throughout the runner.

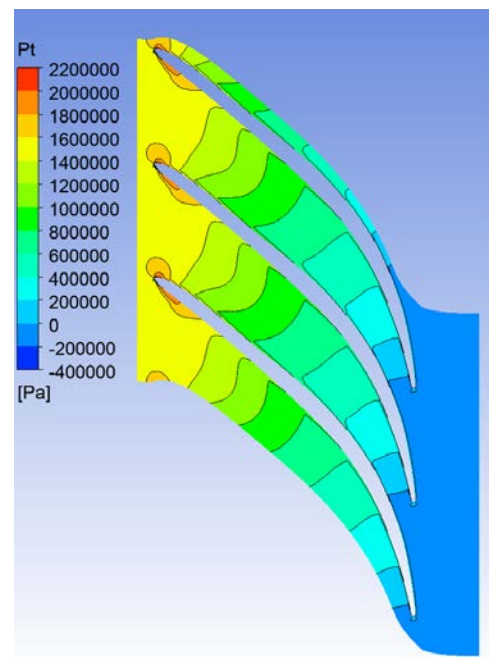


Fig. 4 Pressure contours on the mid plane of the existing blade

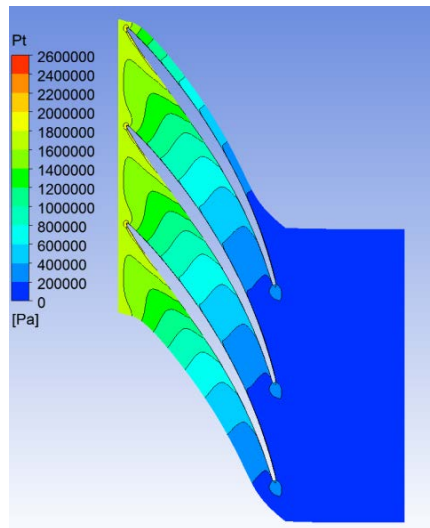


Fig. 5 Pressure contours on the mid plane of the new blade

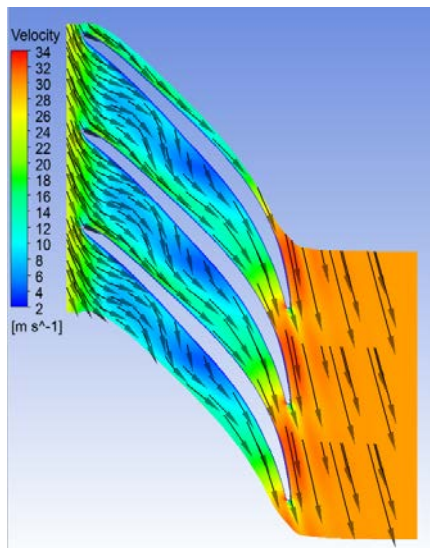


Fig. 6 Velocity vectors on the mid plane of the existing blade

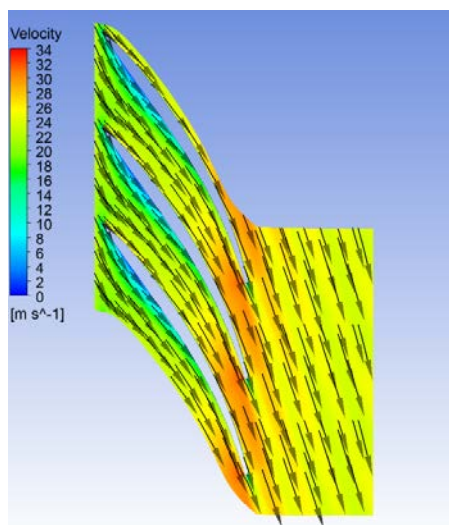


Fig. 7 Velocity vectors on the mid plane of the new blade

Thoma numbers are plotted from the analyses results to observe the cavitation characteristics. As it is seen in Fig. 8, flow cavitates when the existing blades are used, whereas, as Fig. 9 shows cavitation is eliminated in the new design.

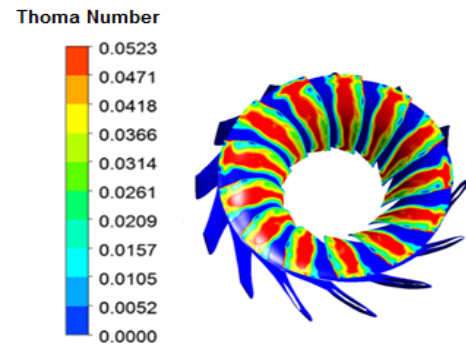


Fig. 8 Thoma number of existing runner

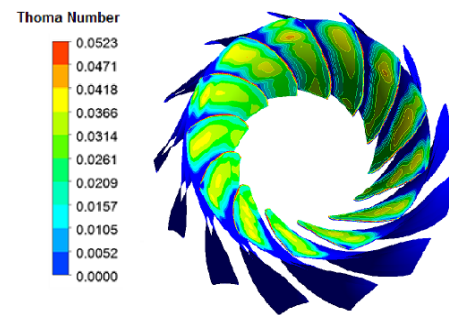


Fig. 9 Thoma number of the new runner

4. Conclusion

In this paper a rehabilitation project for a specific Francis turbine is examined by using CFD tools. The aim of design is to enhance the operating range of the runner, obtain a cavitation free flow and increase the power and the efficiency. According to the results, applied methodology is a successful tool to improve the performance of the runner.

5. Acknowledgments

This study is financially supported by Turkish Scientific and Research Council (Tubitak) under grant 113G109. The computations are performed using the computational facilities of TOBB ETU Hydro Energy Research Lab supported by Turkish Ministry of Development.

6. References

- [1] Okyay G., 2010, "Utilization of CFD tools in the design process of a Francis turbine", M.Sc. Thesis, Middle East Technical University, Ankara, Turkey.
- [2] G. I. Krivchenko, Hydraulic Machines: Turbines and Pumps, Moscow: Mir Publishers, 1986, pp. 22-28.
- [3] Ayli E, Kavurmacı B, Celebioglu K, Aradag S, 2014 "Design and construction of an experimental test rig for hydraulic turbines", 12th Biennial Conference on Engineering Systems Design and Analysis, Copenhagen, Denmark.
- [4] Raabe, J., Hydropower: The Design, Use, and Function of Hydromechanical, Hydraulic, and Electrical Equipment, VDI-Verlag, Verlag des Vereins Deutscher Ingenieure, 1985.
- [5] Wu J., Shimmei K., Tani K., Niikura K., Sato J., 2006, CFD-Based Design Optimization for Hydro Turbines, Journal of Fluids Engineering, 129(2), pp 159-168.
- [6] Odesola, I. F., Oririabre, J. I. 2013, "Development of a 5kw Francis Turbine Runner Using Computation Fluid Dynamics" An International Multidisciplinary Journal, Ethiopia, Vol. 7 (3) pp 178-195.
- [7] Bornard L, Debeissat F, Labrecque Y, Sabourin M, Tomas L, 2014, "Turbine hydraulic assessment and optimization in rehabilitation projects", 27th IAHR Symposium on Hydraulic Machinery and Systems, IOP Conf. Series: Earth and Environmental Science 22 (2014)
- [8] Michel B., Couston M., Sabourin M., Francois M., Hydropower Developments - New Projects, Rehabilitation and Power Recovery, IMechE Headquarters, London, UK, 2004
- [9] Brunes B. T., 2009, "Increasing power output from Francis turbines", M.Sc. Thesis, Norwegian University of Science and Technology
- [10] ANSYS CFX-Pre User's Guide, Release 15.0, November 2013
- [11] ANSYS BladeGen User's Guide, November 2010
- [12] ANSYS TurboGrid User's Guide, Release 15.0, November 2013