CFD BASED DESIGN AND PREDICTION OF PERFORMANCE FOR HYDRAULIC TURBINES

Kutay Celebioglu[1], Burak Altintas[1,2], Yigit Tascioglu[1,2], Selin Aradag[1,2]
[1]Hydro Energy Research Laboratory (ETU Hydro), TOBB University of Economics and Technology, Sogutozu cad No.43 06560 Ankara Turkey
[2]Department of Mechanical Engineering, TOBB University of Economics and Technology, Sogutozu cad No.43 06560 Ankara Turkey
Corresponding author: Burak Altintas, e-mail: burakaltintas@etu.edu.tr

REFERENCE NO

TURB-01

Keywords:
Cavitation
Francis turbine
CFD
Hill chart
Transient effects of flow

ABSTRACT

Every hydroelectric power plant needs a custom hydraulic turbine designed specifically for its flow rate and head characteristics. Turbines must be designed based on these flow parameters considering both design and off-design conditions. This study presents Computational Fluid Dynamics (CFD) based turbine design and prediction of performance and cavitation properties applied to an actual power plant in Turkey: Kepez-1 HEPP in Antalya. The guide vane and runner geometries of this turbine are redesigned with CFD tools in order to obtain higher efficiency and minimize cavitation in its operating range. Hill-chart and cavitation limits are obtained by performing single-phase, steady-state simulations. When compared with the CFD results of the existing turbine, 2.9% efficiency increase reaching 94.48% turbine efficiency at the design point is obtained. To investigate transient effects of flow such as torque fluctuation, rotor-stator interaction and pressure fluctuations, two phase unsteady simulations are performed at the design, part-load and overload operating conditions. Additionally, the steady and unsteady CFD results performed at the same operating conditions are compared, higher efficiency and less cavitation are observed for unsteady simulations because of the fact that frozen rotor interface used for steady-state simulations overestimates flow irregularities. New Kepez-1 turbine can be operated with higher turbine efficiency and better cavitation characteristics in its operating range since unsteady CFD results are more accurate and realistic.

1. INTRODUCTION

Francis turbines are the most common water turbines due to their large head range and higher power capacity. Francis turbines, as other hydraulic turbines, are custom-designed for nominal operating conditions specific to each power plant. They also need to be operated at off-design flow conditions because of variable head and flowrate. Off-design conditions can cause flow irregularities, even cavitation. For instance, part-load operation may cause low frequency pressure fluctuations and power swing due to the cavitating or non-cavitating draft tube vortex. If its frequency coincides with one of natural frequencies of the turbine, strong vibration, noise and surge may take place. Leading edge cavitation, travelling bubble cavitation and inter-blade vortex cavitation occur at different operating conditions, which may cause erosion, reduction in efficiency, vibration and noise [1]–[7]. To avoid these cavitation types or minimize their undesirable effects, the proper design of turbine components and determining cavitation limits are vital.

By the help of rapidly developing computer technology, Computational Fluid Dynamics (CFD), having outstanding abilities and accuracy, became an irreplaceable tool in design, examining performance and cavitation properties. In this study, the guide vane and runner geometries of Kepez-1 Francis turbine are redesigned with CFD tools in order to obtain higher efficiency and minimize cavitation in its operating range. The performance and cavitation properties of the new Kepez-1 turbine in its operating range are investigated in detail by performing single phase steady-state simulations. Two phase transient simulations at the design, part-load and overload operating points are also performed to investigate the transient effects of flow such as torque fluctuation, rotor-stator interaction and pressure fluctuation.

2. THE DETAILS OF THE NEW DESIGN

Kepez-1 HEPP, is located in Antalya Turkey, is designed in 1957 and has been operated for sixty years. Therefore, the rehabilitation of the turbine is necessary in order to increase turbine efficiency at both design and off-design conditions, and minimize cavitation. For the new turbine, at least 94% turbine efficiency is in demand for the design condition (160 m, 6.1 m^3/s). The geometries of runner and guide vane of Kepez-1 turbine are redesigned using CFD.

Kepez-1 HEPP consists of three vertical Francis turbines. Technical specifications of the turbines are shown in Table 1.

Table 1.Technical specifications of Kepez-1 turbine

Net head (m)	160
Discharge (m^3/s)	6.1
Rev. speed (rpm)	750
Setting level (m)	1.45
Runner diameter (m)	1.234
# of stay vanes	10
# of guide vanes	20
# of runner blades	13

The geometry of new runner is generated by using ANSYS BladeGen. The meridional profile, blade thickness, θ and δ angles are redesigned by performing many single runner blade simulations using CFD tools iteratively. The new runner shown in Fig. 1 is obtained. The existing runner is scanned by laser scanner, so that geometrical features of both designs can be compared.

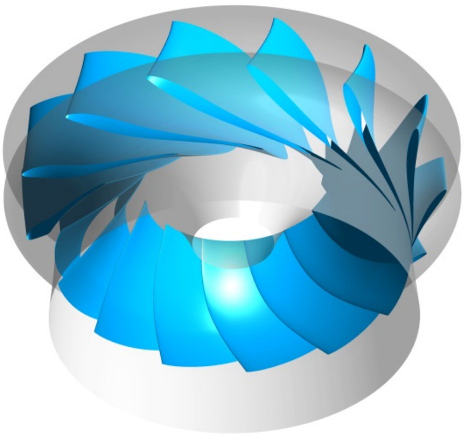

Fig. 1.New runner geometry

The meridional profile of the existing runner (red line) is changed as shown in Fig. 2. The trailing edge curve of new profile is approached to leading edge, so the blade length is decreased significantly in the new design.

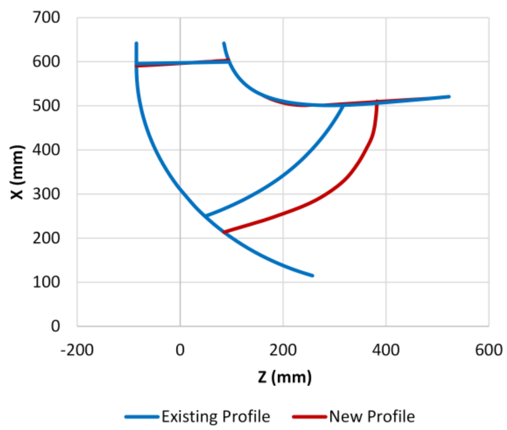

Fig. 2. Meridional profiles of the existing and new runner.

The new runner is designed by using X-blade runner technology. This technology provides less cavitation, higher efficiency, higher operation stability at wider operating range [8], [9].

Design parameters for the guide vanes are the thickness and guide vane angle. Thus, NACA0016 profile and 23^o guide vane angle providing 6.1 m^3/s flowrate and 16.5^o incidence angle to new runner are determined by performing several turbine simulations using CFD tools iteratively.

3. NUMERICAL METHODOLOGY

Total number of mesh elements is around 26 million for the CFD analysis of the turbine.

Table 2 shows the mesh properties for each part of the turbine.

Table 2. Mesh properties

Components	Type of elements	# of elements	y^+_{ort}
Spiral case	Tetrahedral	8013242	41
Guide vanes	Hexahedral	6471280	107
Runner	Hexahedral	8504106	110
Draft tube	Hexahedral	3176566	46
	Total	26165194	

All simulations are carried out by using the commercial code ANSYS CFX 15.0. Shear Stress Transport (SST) is used as turbulence model. The boundary conditions used are total pressure at the inlet of the spiral case and static pressure at the outlet of the draft tube. For wall roughness, smooth wall is used.

Single phase, steady-state simulations are performed for sixty different operating points ranged from 140 m to 175m head and from 14^o to 29^o guide vane opening angle as shown in Fig. 3. Frozen rotor interface is used as rotor-stator interface.

Two phase, transient simulations are performed at the design, part-load and overload operating points seen in Fig. 3 as blue, red and yellow stars, respectively. Rayleigh Plesset cavitation model is implemented for the detection of cavitation. Transient rotor-stator interface is used. A second order backward Euler scheme is used for temporal discretization. The time step is selected as 2^o degrees of runner revolution.

All simulations are performed with ETU Hydro HPC cluster, which comprise nine HP Proliant DL380P Gen8 compute nodes, each having 12 cores. While the sixty steady simulations take 15 days to run, three unsteady simulations take 36.5 days. Thus, all simulations take 51.5 days.

4. RESULTS

Steady-state simulations are performed for sixty operating points. The hill chart representing the performance for the operating range is shown in Fig. 3. According to the results presented in this chart, the turbine can be operated with 94.48% efficiency at the design point, 90% or more turbine efficiency for the whole operating range. Cavitation analysis is performed in detail for each simulation to determine the cavitation limits of the turbine. They indicate that the turbine does not suffer leading edge and inter-blade vortex cavitations for any operating point. Travelling bubble cavitation occurs at the operating area having more flowrate from the purple line, in addition, draft tube vortex cavitation occurs at the operating area having more flowrate from the green line. Thus, new Kepez-1 turbine can be operated without cavitation in the operational range (red area), on the other hand, it can be operated 100 hours per year in the temporary operational range (claret red area) according to IEC 60609 [10].

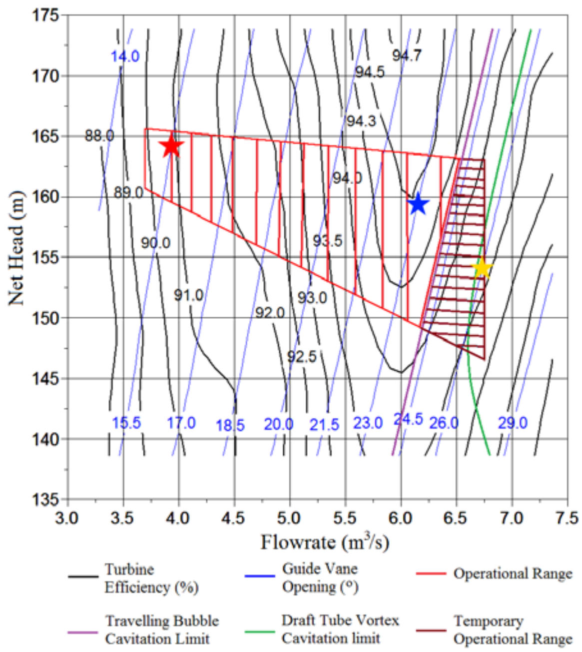

Fig. 3. Hill-chart of the new Kepez-1 turbine

In this study, the CFD results of simulations at the design point (160 m head, 23^o opening), part-load (165 m head, 15.5^o opening) and overload (155 m head, 26^o opening) are investigated.

4.1. Design Point

At the design point, the turbine efficiency is obtained as 94.48%. When compared with the CFD results of the existing turbine, a 2.9% increase is provided in the new design.

When the steady and unsteady CFD results seen in Table 3 are compared, the unsteady simulation predicts 0.1 m^3/s lower flowrate, 0.8% higher efficiency and 59.6 kW lower shaft power.

Table 3. Performance values at design point

	Steady	Unsteady
Flowrate (m^3/s)	6.14	6.04
Net Head (m)	158.73	158.72
Turbine Eff. (%)	94.48	95.30
Power (kW)	8995.81	8936.20

As seen in Fig. 4, the torque fluctuation is nearly zero, hence it can be said that there is no power swing.

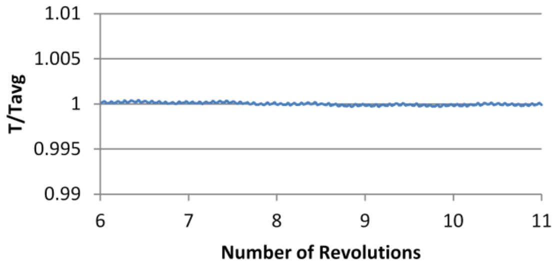

Fig. 4. Torque fluctuation at design point

4.2. Part-load operating condition

When the steady and unsteady CFD results seen in Table 4 are compared, the performance values are nearly same with each other. Unsteady simulation predicts 40 kW lower shaft power corresponding to 0.03 m^3/s decrease in flowrate.

Table 4. Performance values at part-load

	Steady	Unsteady
Flowrate (m^3/s)	3.93	3.90
Net Head (m)	163.81	163.78
Turbine Eff. (%)	90.83	90.82
Power (kW)	5712.73	5672.68

As seen in Fig. 5, the peak to peak torque fluctuation is about 1.6%. This means that there is a 91 kW power swing. This swing is caused by low frequency (0.33 f_n) pressure fluctuations due to the non-cavitating helical vortex core seen in Fig. 6b.

In Fig. 6b, the helical vortex core appears greater, because unsteady simulations predict the α angle in the runner outlet lower as seen from the streamlines in Fig. 6, so the circumferential velocity (v_u) increases and the vortex core becomes more.

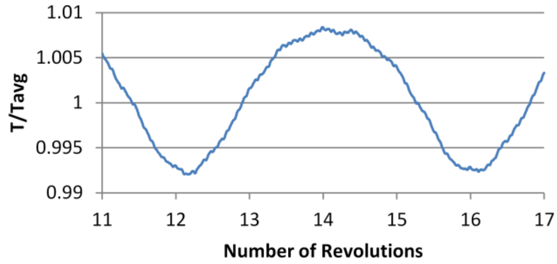

Fig. 5. Torque fluctuation at part-load

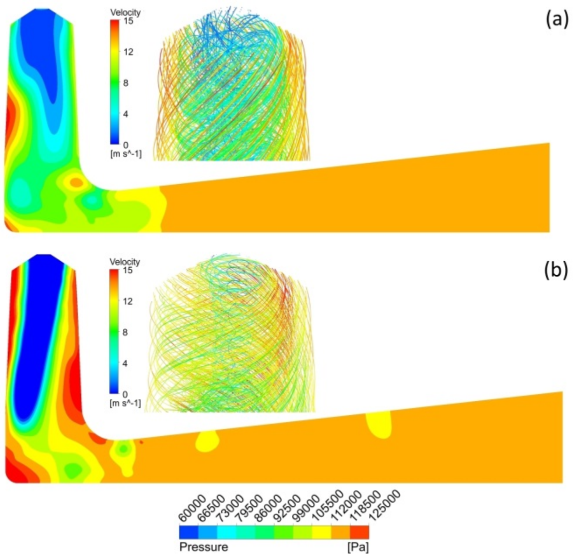

Fig. 6. Pressure contours on the draft tube for steady-state (a), and unsteady-state (t=1.36 s) (b)

4.3. Overload operating condition

When the steady and unsteady CFD results seen in Table 4 are compared, the unsteady simulation predicts 0.13 m^3/s lower flowrate, 1% higher efficiency and 92.1 kW lower shaft power.

Table 5. Performance values at overload

	Steady	Unsteady
Flowrate (m^3/s)	6.72	6.59
Net Head (m)	153.69	153.66
Turbine Eff. (%)	93.65	94.68
Power (kW)	9459.92	9367.79

As seen in Fig. 7, the torque fluctuation is nearly zero, hence it can be said that there is no power swing.

Fig. 8a shows travelling bubble cavitation. However, no cavitation appears at the results of the unsteady simulation as seen in Fig. 8b. There are two reasons for this difference. The first one is that frozen rotor interface used for steady simulations overestimates flow irregularities [11], [12], so this causes less cavitation in the unsteady results. The second one is that the flowrate is predicted lower in the unsteady results. Travelling bubble cavitation is highly dependent on flowrate because it appears at overload operating conditions [1]–[3].

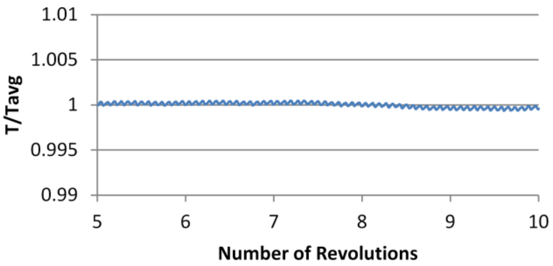

Fig. 7.Torque fluctuation at overload

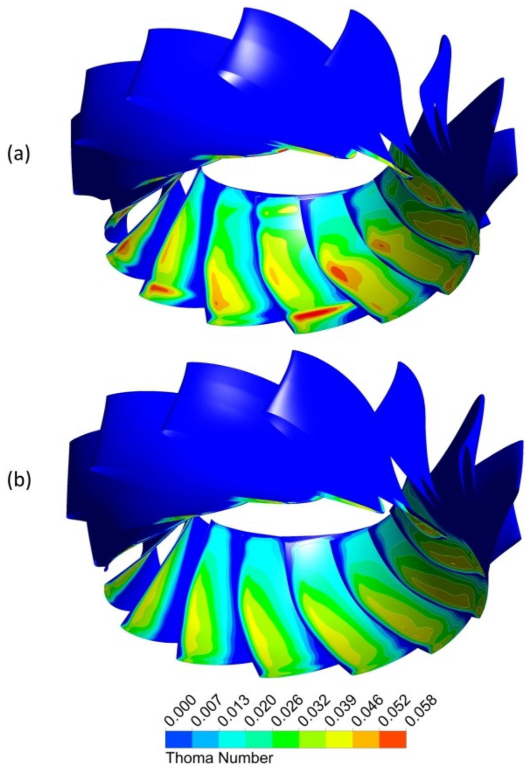

Fig. 8.The Thoma contours on the runner for steady-state (a), and unsteady (t=0.8s) (b)

5. CONCLUSIONS

In this study, the guide vane and runner geometries of Kepez-1 Francis turbine are redesigned with CFD tools in order to obtain high efficiency and minimize cavitation in its operating range. When compared with the CFD results of the existing turbine, a 2.9% efficiency increase is obtained reaching 94.48% turbine efficiency at the design point, and the performance targets are accomplished by providing 90% or more turbine efficiency for the entire operating range. The result of steady state simulation shows that travelling bubble cavitation is observed at the overload operating condition. The same cavitation properties are not observed when two-phase, unsteady simulation is performed at the same operating condition. Additionally, turbine efficiency is predicted higher. This is because frozen rotor interface used for steady-state simulations overestimates flow irregularities. In the results of unsteady simulations, this causes less cavitation and hydraulic losses, therefore higher turbine efficiency is obtained. New Kepez-1 turbine can be operated at its design point with 95.3% turbine efficiency. When compared with the results of steady-state simulations, it has higher turbine efficiency and better cavitation performance in its operating range since unsteady CFD results are more accurate and realistic.

The new runner and guide vanes are manufactured in model scale. Performance and cavitation tests of the reduced scale model of the new Kepez-1 turbine are currently being performed at ETU Hydro Test Rig in order to validate the numerical results obtained in this study.

Fig. 9. Runner model

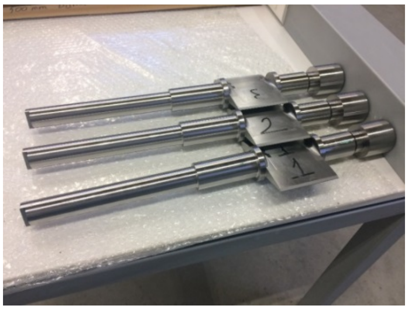

Fig. 10. Guide vane model

Acknowledgements
This work is financially supported by Scientific and Technological Research Council of Turkey (TUBITAK) under grant 113G109. The facilities of TOBB ETU Hydro Energy Research Center (ETU Hydro) financially supported by Turkish Ministry of Development, are utilized.

Nomenclature

f_n Non-dimensional pressure fluctuation frequency by runner rotational frequency n [13].

Greek Letters

θ Azimuth angle (degrees)

δ Blade angle (degrees)

References

[1] S.C. Li, Ed., *Cavitation of Hydraulic Machinery*. Imperial College Press, 2000.

[2] X. Escaler, E. Egusquiza, M. Farhat, F. Avellan, and M. Coussirat, “Detection of cavitation in hydraulic turbines,” *Mech. Syst. Signal Process.*, vol. 20, no. 4, pp. 983–1007, 2006.

[3] F. Avellan, “Introduction to cavitation in hydraulic machinery,” in *The 6th International Conference on Hydraulic Machinery and Hydrodynamics Timisoara*, 2004, pp. 11–22.

[4] K. Celebioglu, B. Altintas, S. Aradag, and Y. Tascioglu, “Numerical research of cavitation on Francis turbine runners,” *Int. J. Hydrogen Energy*, vol. 42, no. 28, pp. 17771–17781, 2017.

[5] P. Kumar and R. P. Saini, “Study of cavitation in hydro turbines-A review,” *Renew. Sustain. Energy Rev.*, vol. 14, no. 1, pp. 374–383, 2010.

[6] A. KC, Y. H. Lee, and B. Thapa, “CFD study on prediction of vortex shedding in draft tube of Francis turbine and vortex control techniques,” *Renew. Energy*, vol. 86, pp. 1406–1421, 2016.

[7] D. Valentín, A. Presas, E. Egusquiza, C. Valero, M. Egusquiza, and M. Bossio, “Power Swing Generated in Francis Turbines by Part Load and Overload Instabilities,” *Energies*, vol. 10, no. 12, p. 2124, 2017.

[8] J. T. Billdal, “The X factor.” [Online]. Available: http://www.waterpowermagazine.com/features/featurethe-x-factor/. [Accessed: 17-Oct-2017].

[9] H. Brekke, “A Review on Oscillatory Problems in Francis turbines,” *New Trends Technol. Devices, Comput. Commun. Ind. Syst.*, pp. 217–232, 2010.

[10] IEC 60609, “Hydraulic turbines , storage pumps and Cavitation pitting evaluation - Part 1: Evaluation in reaction turbines, storage pumps and pump-turbines,” 2005.

[11] H. Keck and M. Sick, “Thirty years of numerical flow simulation in hydraulic turbomachines,” *Acta Mech.*, vol. 201, pp. 211–229, 2008.

[12] “Best practice guidelines for turbomachinery CFD.” [Online]. Available: https://www.cfd-online.com/Wiki/Best_practice_guidelines_for_turbomachinery_CFD. [Accessed: 24-Oct-2017].

[13] IEC 60193, “Hydraulic turbines, storage pumps and pump-turbines – Model acceptance tests,” 1999.