OPTIMIZATION OF A MODEL FRANCIS TURBINE'S PARAMETERS FOR THE MOST EFFICIENT PERFORMANCE CASE

Deniz Sarper SEMERCİ, Tahir YAVUZ
Mechanical Engineering Department, Başkent University, Ankara/Turkey
Corresponding author: Deniz Sarper SEMERCİ, e-mail: dssemerci@baskent.edu.tr

REFERENCE NO

TURB-02

Keywords:
Francis Turbine, CFD, Performance, Taguchi

ABSTRACT

The significance of renewable energy systems has been increasing day by day due to insufficient fossil sources and clean energy demands of countries. Most of the countries are inclining to different renewable energy sources such as hydraulic energy. Hydraulic energy has a high potential in all over the world. In this study, a model turbine which is designed based on an actual turbine operating at Kesikköprü Hydroelectric Power Plant is considered. The aim of the study is to determine the optimum blade angle values leading the runner to provide maximum performance by using Taguchi optimization technique and Computational Fluid Dynamics analyses.

1. INTRODUCTION

The energy demand of the countries is increasing evenly with the increment of human population. Countries were supplying their energy demands by fossil fuels, previously. Limited resources and effects on environment of fossil fuels led countries to search new energy resources. In recent decades, countries are trying to generate new and sustainable solutions for energy demands. In this context, renewable energy becomes the most important energy solution to supply the energy demands. Renewable energy is the energy acquired from natural processes that are renovated faster than they are consumed. There are several types of renewable energy sources such as solar energy, wind energy, geothermal energy, hydraulic energy, etc. Hydraulic energy has an importance among them because of its high potential in all over the world and it is efficiently transformed into electrical energy. Thus, hydraulic energy has a remarkable importance among these renewable energy sources on supplying energy demands of countries. The Hydraulic energy potential of Turkey is; 433 billion kWh theoretical potential, 216 billion kWh technical potential, 140 billion kWh economical potential and 13000 MW installed capacity. The aim of the Ministry of Energy is to increase the capacity to 36000 MW in 2023 [1]. To reach this target, efficiency is the most important parameter in turbomachines that are used in hydroelectric power plants. Increasing the efficiencies of these turbomachines provides more energy to acquire.

Hydraulic turbines are the turbomachines that using kinetic and potential energy of fluid to generate electrical energy. There are two types of hydraulic turbines; action and reaction turbines. The difference between these two types are pressure difference between inlet and outlet of the turbine. Francis turbine is a reaction turbine which is accepted as the most efficient one among the hydraulic turbines [2]. In Turkey, Francis turbine is using widely in hydroelectric power plants. But many of these Francis turbines in use were built far-back. And these turbines' efficiencies have been diminishing due to many reasons such as cavitation, fauling etc. For more efficient electric energy generation, these turbines' efficiencies should be increased by rehabilitation processes. As long as new technologies and techniques are found out, these turbines can be modified and make them to generate more energy due to modifications and new tools.

Francis turbines are the most common hydraulic turbine type in use. A Francis turbine comprises of five main components: spiral case, stay vanes, guide vanes, runner and draft tube. A schematic view of Francis turbine is given in Figure 1.

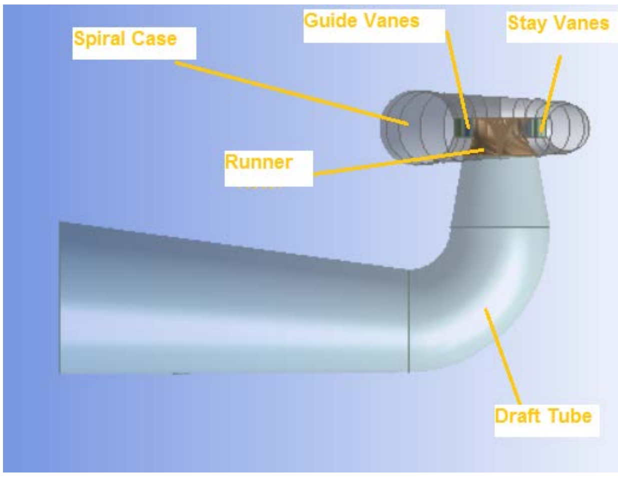

Fig. 1. Francis Turbine Components

Water first enters the spiral case which is the component that provides the pressure to spread equally to the vane sections. Spiral case's decreasing cross sectional area provides this equal pressure distribution on each section. Stay vanes are the flow regulators that provides flow to enter properly to the guide vanes. Guide vanes are the component that controls the flow rate of the Francis turbine by changing their angles, by servomotors generally. Guide vanes are also send water to the runner blades with uniform streamlines. That means guide vanes are regulating the flow through the runner. Water then proceeds on its way to the runner. Water particles hits the turbine blades and turbine runner starts to rotate dependently. A shaft is located in the middle of the runner which is rotating as water rotates the runner. This shaft is engaged to a generator and this shaft drives the generator to generating electric energy. Water should be discharged from runner area after rotating the runner. Draft tube is located at the outlet section of runner and water that rotates the runner is falling out to this component and goes along it. Draft tube increases waters pressure and decreases its velocity by using its increasing cross sectional area and serves flow to the discharge canal at atmospheric pressure. Thus, a cycle of Francis turbine is completed after water leaves the draft tube.

The aim is; using optimization techniques and Computational Fluid Dynamics analyses to determine the optimum blade angles leading the maximum efficiency of the runner.

1.1. Literature Survey

There are some studies in literature about Francis turbines, Taguchi optimization techniques and Computational Fluid Dynamics (CFD).

Francis turbine is invented by James B. Francis at 1855.

Schweiger [3] examined the dimensionless parameters, geometrical quantities and specific speed of a Francis turbine. Siervo and Leva [4] manifested diagrams and equations about the design of the spiral case, runner and draft tube. Nilsson and Davidson [5] developed a CFD code for solving turbulent flow in complex domains of a water turbine. Teran et al. [6] investigated the design of guide vane blade geometry by using genetic algorithm in order to send the water in desired angle and flow rate to the runner and they examined the velocity and pressure distributions. Khan [7] examines the cavitation phenomena in Francis turbine runner and make some suggestions the prevent cavitation in turbine runner. Anup et al. [8] underlined the effect of vortex structure at draft tube and they obtained the pressure fluctuations along the draft tube. Ayancık and Çelebioğlu [9] examined turbine performance, velocity and pressure distributions of a Francis turbine by CFD codes and they obtained the most influent parameters on Francis turbine performance.

Kelkar investigated the optimization of the cooling rate of the turbine blades by using Taguchi technique and determined that "In Taguchi technique, the word "optimization" implies determination of best levels of control factors [10].

2. METHOD

A Design methodology is created. First step is specifying head and discharge values that are taken, and scaled based on affinity laws, from an actual turbine at Kesikköprü Hydroelectric Power Plant. Then, preliminary design is made based on theoretical data on literature. Geometry design is made by using computer aided design programmes and computational domains of Francis turbines are generated. Then, Taguchi technique is applied to determined blade angles and taken a experimental design from this technique. Each experimental design needs a response value to realize the analyses. The responses are selected as torque values from CFD analyses. So, CFD analyses are the next step. Based on CFD analyses response values are linked to the Taguchi analysis and finally, Taguchi analysis gives us the optimal values of each angles. Based on this given optimal values another CFD analysis is carried out for verification of Taguchi analysis. Then the final design is determined.

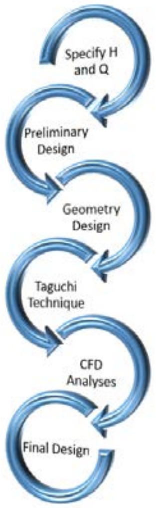

Fig. 2. Design Methodology

2.1. Taguchi Analysis

Taguchi technique is an experimental design optimization technique which uses standard orthogonal arrays for forming a matrix of experiments in such a way as to extract maximum important information with minimum number of experiments [10].

MINITAB 17.0 is used for Taguchi analysis. Three level analyses are carried out for three factor Taguchi designs. Three factors are defined as guide vane angle which is referred by factor A, runner inlet angle which is referred by factor B and runner outlet angel which is referred by factor C. The angle values are calculated by using technical data and experimental curves from literature. The factor values are specified by selecting nearest high and low values of the calculated values with equal intervals.

Taguchi algorithm suggests two possible designs: L9 and L27. The numbers that comes after the letter L are indicates the experiment number that should be carried out. For saving time and computational rams, the L9 design is selected. That means nine experiments should be done for specifying optimal factor levels. The L9 Taguchi design is given in table below. CFD analysis are carried out for nine different designs.

Table 1: L9 Taguchi Design

Experiments/Factors	A	B	C
1	20	60	20
2	20	70	22
3	20	80	24
4	22	60	22
5	22	70	24
6	22	80	20
7	24	60	24
8	24	70	20
9	24	80	22

Hydraulic turbine efficiency is defined as the ratio of shaft power to hydraulic power that depends on head and flow rate values. Since the head and flow rate values are constant for this model turbine, the torque value is the main parameter that directly effects the turbine efficiency. So, torque values are obtained from analyses and entered as response column for these nine different experimental designs. And Taguchi analysis is carried out.

$$\eta_{turb.} = \frac{P_{shaft}}{\rho g Q H} = \frac{T\omega}{\rho g Q H} \qquad (1)$$

2.2. CFD Analysis

Computational Fluid Dynamics (CFD) codes are widely common tools in use to predict turbomachinery performance in recent years. In this study, ANSYS 18.1 Academic version is used for analysis. ANSYS Design Modeler is used for generating flow domains of Francis turbine, ANSYS Mesher is used for mesh structures of the flow domain and ANSYS CFX is used for solving Reynolds Averaged

Navier-Stokes(RANS) equations for specified mesh structures. The using RANS equations by the commercial CFD code is given below in general form [11].

$$\rho \frac{D\bar{u}}{Dt} = \rho B_x - \frac{\partial \bar{p}}{\partial x} + \frac{\partial}{\partial x}\left[\mu \frac{\partial \bar{u}}{\partial x} - \rho \overline{u'^2}\right] + \frac{\partial}{\partial y}\left[\mu \frac{\partial \bar{u}}{\partial y} - \rho \overline{u'v'}\right] + \frac{\partial}{\partial z}\left[\mu \frac{\partial \bar{u}}{\partial z} - \rho \overline{u'w'}\right] \quad (2)$$

$$\rho \frac{D\bar{v}}{Dt} = \rho B_y - \frac{\partial \bar{p}}{\partial y} + \frac{\partial}{\partial x}\left[\mu \frac{\partial \bar{v}}{\partial x} - \rho \overline{u'v'}\right] + \frac{\partial}{\partial y}\left[\mu \frac{\partial \bar{v}}{\partial y} - \rho \overline{v'^2}\right] + \frac{\partial}{\partial z}\left[\mu \frac{\partial \bar{v}}{\partial z} - \rho \overline{v'w'}\right] \quad (3)$$

$$\rho \frac{D\bar{w}}{Dt} = \rho B_z - \frac{\partial \bar{p}}{\partial z} + \frac{\partial}{\partial x}\left[\mu \frac{\partial \bar{w}}{\partial x} - \rho \overline{u'w'}\right] + \frac{\partial}{\partial y}\left[\mu \frac{\partial \bar{u}}{\partial y} - \rho \overline{v'w'}\right] + \frac{\partial}{\partial z}\left[\mu \frac{\partial \bar{u}}{\partial z} - \rho \overline{w'^2}\right] \quad (4)$$

2.2.1. Mesh generation

The flow domains of Francis turbine are separately meshed in ANSYS Mesher. The mesh element shapes are specified as tetrahedral and hexahedral.

Face and body sizing options are used to obtain more sensitive results. More elements are used around the blade profiles of stay vanes, guide vanes and runner blades due to criticality of these regions. Proximity and curvature size function is used for generating an automatic refinement on critical element intersections. The obtained skewness values are acceptable for each component. Besides, the skewness value of runner is a bit higher due to its complex structure with respect to other components.

All meshed flow domains are assembled for analysis and shown in Figure below.

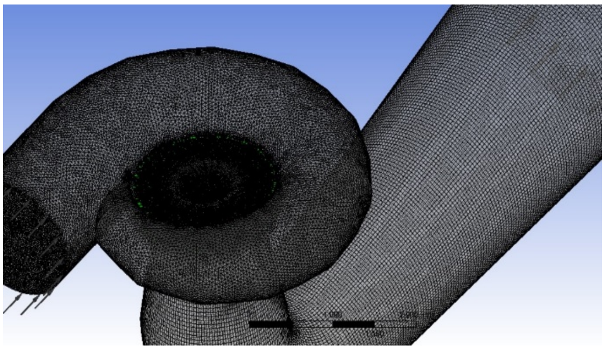

Fig. 3. Mesh Structure

The mesh structure is controlled in different types of mesh model for grid-independency and it is stated as independent from mesh.

2.2.2. Models and approaches used in CFD analysis

In this study, ANSYS CFX 18.1 is used as commercial RANS codes for modelling turbulent flow. The commercial code is using finite volume method as discretization scheme. RNG k-ε is used as turbulence model due to its robustness. High resolution option is selected for advection scheme and the convergence criteria is specified as 10^{-5} for k, ε and velocity residuals.

Mass flow inlet is defined to the spiral case inlet and static pressure outlet is defined to the draft tube outlet as boundary conditions. The reference pressure is defined as 0 atm.

Because of assembling all the components in CFX, there is a need for some approaches to interface regions of these components. Due to the moving structure of runner and stable structure of guide vanes and draft tube, there should be need for applying some approaches on these components' interface regions. Multiple Reference Frame (MRF) method is used for modelling interfaces. Frozen Rotor is used on the interfaces between runner and guide vanes and runner and draft tube. General Grid Interface is used for the other interface regions.

3. RESULTS AND DISCUSSIONS

Nine different CFD analyses are carried out by using ANSYS CFX 18.1 based on the experimental design combinations that obtained from Taguchi algorithm. The torque values that obtained from CFD analyses are entered into MINITAB 17.0 as the response values and thus the setup is completed and Taguchi analysis is carried out. Response tables for means, S/N values and Anova table is obtained from Taguchi analysis.

Table 2: Response Table for Means

Level	A	B	C
1	13718	20170	20316
2	16617	20412	20275
3	30554	20306	20298

Delta	16836	242	41
Rank	1	2	3

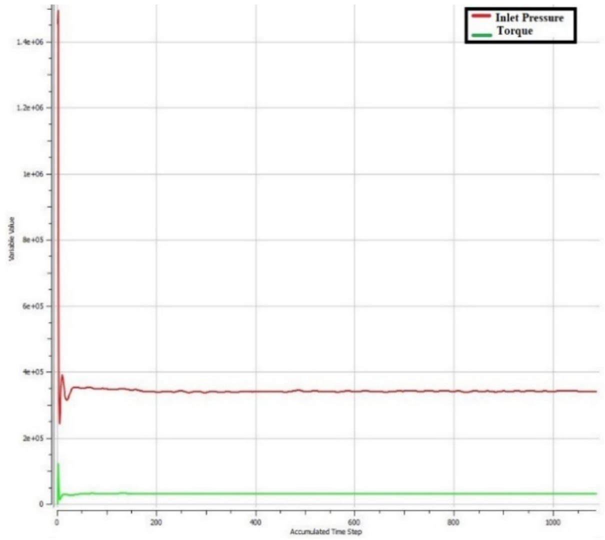

Fig. 4. Torque and Inlet Pressure values depend on iterations

The main effects for means and plot for S/N ratios are shown in Figure. It can be seen that the plot characteristics are entirely same with the plot for means due to selecting “Larger the better” approach.

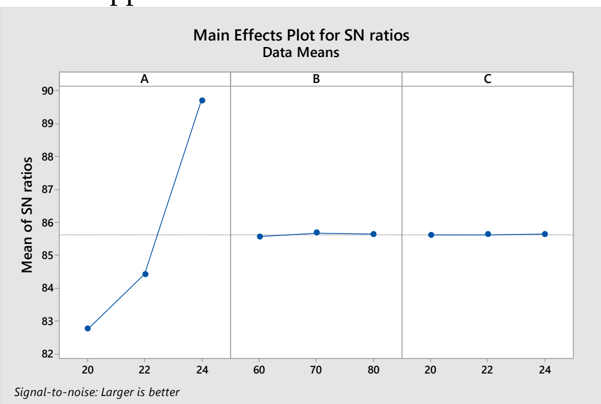

Fig. 5. Main Effects Plot for S/N Ratios

For factor A, the optimal level is determined as 3. For factor B, the optimal level is determined as 2 and for factor C, the optimal value is determined as 3. That means the optimum guide vane angle is 24°, the optimum runner inlet angle is 70°, and the optimum runner outlet angle is 24°.

Table 3: Response Table for Signal to Noise Ratio

Level	A	B	C
1	82,40	85,56	85,61
2	84,41	85,67	85,61
3	89,70	85,63	85,63
Delta	6,96	0,11	0,02
Rank	1	2	3

Fig. 6. Main Effects Plot for Means

The response tables for means and S/N are given in Table 2 and 3. It also can be seen form these tables that the optimal values are 3 for factor A, 2 for factor B and 3 for factor C. After obtaining response tables for means and signal to noise values, an Anova table is obtained. The importance of Anova table is to determine the most effective parameter on turbine efficiency. This determination is done in terms of “P value” which located in the right hand column in Anova table. The critical P value is generally selected as 0,05 in literature. That means, if the factor’s corresponding P value is smaller than the critical P value, this factor is effective on turbine efficiency.

Table 4: P-Values from Anova Table

Factors	P-Value
A	0,000
B	0,018
C	0,823

It can be seen that factor A which is referred to guide vane angle is the most effective parameter on turbine analysis. According to corresponding P values, the runner inlet angle is the second effective parameter on turbine efficiency. It can be seen that the P value of the factor C which is referred runner outlet angle is not quite effective on turbine’s torque and efficiency with respect to other parameters.

After Taguchi analyses are done and the algorithm suggests the optimal values of each factors, it’s seen that the optimum design combination that is obtained from algorithm does not exists in first nine design combinations. Thus, a verification analysis is carried out for the values of guide vane angle 24˚, runner inlet angle 70˚ and runner outlet angle 24˚. The torque value is obtained as previous analyses and it's seen that the optimal design which is suggested by Taguchi has the highest torque value among these nine design combinations. So, it can be said that the Taguchi analysis is carried out correctly and the results are realizable.

Nine different CFD analysis for each case and a verification analysis are carried out on ANSYS CFX. Pressure distributions, velocity distributions, streamlines are obtained for each combinations. The verification analysis’ results are considered because of the optimal values are obtained in these combination of factors. In Figure 7, pressure distribution of the spiral case is shown. Vectors show that the flow is regular on each section of spiral case and the pressure distribution shows that spiral case is spreading the pressure equally on its own sections because of its decreasing cross sectional area.

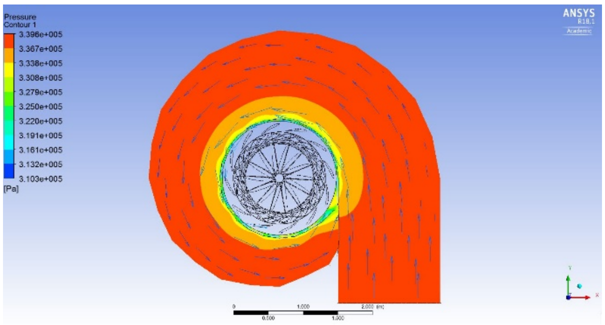

Fig. 7. Pressure distribution and velocity vectors on spiral case

In figure 8, pressure distribution of guide vanes and velocity vectors are shown. It can be seen that water flows regularly among the guide vanes. Considering velocity vector lines’ colour, white represents the low velocity and blue represents the higher velocity. Thus, the flow velocity is increasing through the runner. That means guide vanes are controlling the flow velocity and the flow rate of the Francis turbine.

So, considering the P-value of guide vane angle on Taguchi results and the velocity vectors on CFD results, it can be said that guide vanes are the most important component on controlling turbine efficiency.

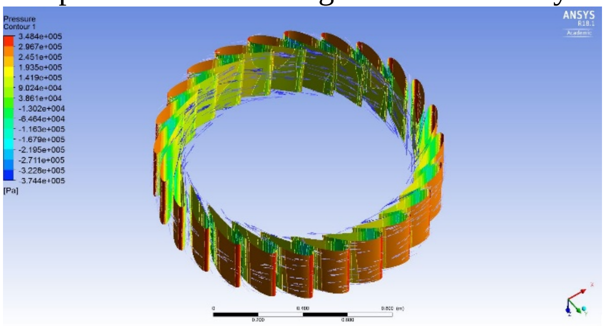

Fig. 8. Pressure distribution and velocity vectors on guide vanes

It can be said that, water is flowing regularly based on stagnation point’s location on the symmetry point of guide vane leading edge.

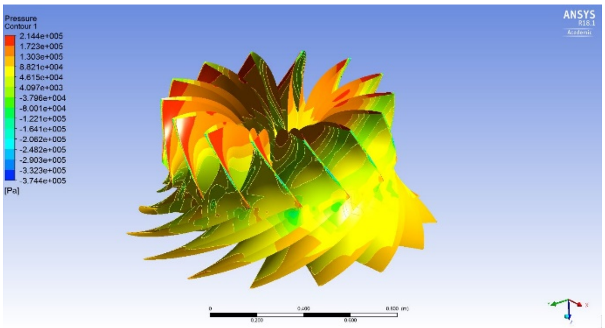

Fig. 9. Pressure distribution of runner

Figure 9 shows that pressure and suction sides of runner blades are exists correctly to move turbine runner. It can be seen that the pressure is decreasing through the runner outlet section due to the increase in velocity.

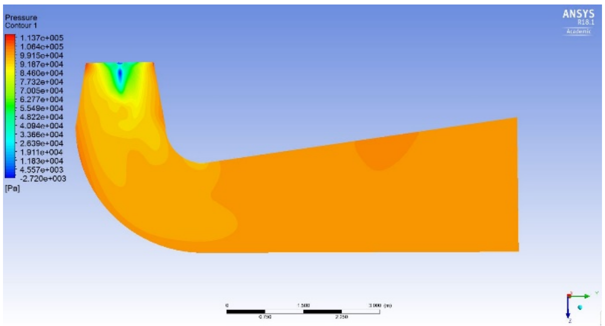

Fig. 10. Pressure distribution of draft tube

The draft tube pressure distribution is shown in Figure 10. Pressure is increasing gradually through the outlet section of draft tube and it exits draft tube at atmospheric pressure, 101325 Pa. The flow is quite regular besides the elbow effects of the runner outlet corners. The pressure fluctuations are as few as possible as it can be seen in the same figure.
The streamlines of the water flow in Francis turbine is shown in Figure 11. Velocity increment starts from guide vanes and water enters as fast as possible to the runner blades.

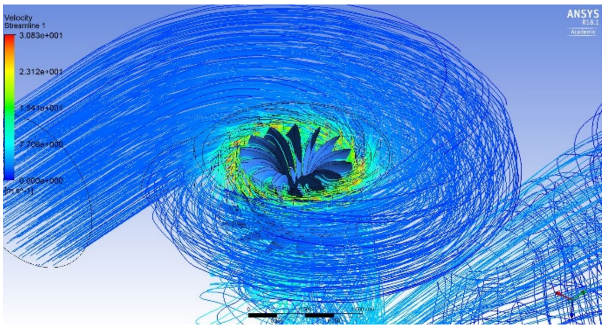

Fig. 11. Flow streamlines in Francis turbine

After rotating runner, water leaves runner with a high velocity and draft tube absorbs its velocity by its increasing cross sectional area and increase flow’s pressure to the outlet section.
Consequently, it can be said that the obtained pressure and velocity distributions are approved as appropriate.

4. CONCLUSION

A Francis turbine is designed based on an actual turbines head and discharge values. A scaling process is applied, due to affinity laws, on to this actual head and discharge and diameter values and a model turbine geometry is generated based on these scaled parameters.
Computational domains of this Francis turbine geometry is designed in ANSYS Design Modeler. Taguchi analysis is made to have an experimental design and based on this experimental design, CFD analyses are carried out by using ANSYS CFX 18.1. The torque values that obtained from CFD analyses are determined as the response values for Taguchi technique and Taguchi analysis is done. Due to the results of Taguchi analysis, the optimal blade angles are specified as 24 ˚ for guide vanes, 70 ˚ for runner inlet angle and 24 ˚ for runner outlet angle.
A verification analysis is carried out based on these optimal values and it is seen that the obtained torque value is higher than other nine experimental designs. That means Taguchi analysis is done correctly. So it can be said that the optimal blade angle values are determined accurately for this model Francis turbine.

Nomenclature

A	Taguchi factor that represents guide vane angle (˚)
B	Taguchi factor that represents runner inlet angle (˚)
C	Taguchi factor that represents runner outlet angle (˚)
g	Gravitational acceleration (m/s^2)
H	Turbine head value (m)
Q	Turbine discharge value (m^3/s)
u	Velocity in x direction (m/s)
v	Velocity in y direction (m/s)
w	Velocity in z direction (m/s)

Greek Letters

η_{turb}	Turbine efficiency (%)
P_{shaft}	Shaft power (W)
ρ	Density (kg/m^3)
μ	Dynamic viscosity (kg/ms)

References

[1] http://www.eie.gov.tr/yenilenebilir/h_turkiye_potansiyel.aspx
[2] Y. Çengel, J.M. Cimbala, Fluid Mechanics Fundamentals and Applications, 2nd edition McGrawHill, 2010.
[3] Schweiger, F., Gregori, J., Developments in Francis Turbines, Water Power and Dam Cont., Aug. 1985, 17-21.
[4] Siervo, J., Leva, F., Modern Trends in Selecting and Designing Francis Turbines, Water Power and Dam Cont., Aug., 1976, 28-35.
[5] Nilsson, H., Davidson, L.A., Numerical Comparison of Four Operating Conditions in a Kaplan Water Turbine, Focusing on Tip Clearence Flow Published in the Proceedings of the 20th LAHK

Symposium, Charlotte, North Caroline-USA, Aug. 2000.
[6] Teran, L.A., Larrahondo, F.J., Rodriguez, S.A., Performance Improvement of a 500-kW Francis turbine based on CFD, Renewable Energy, 96 , 2016, 977-992.
[7] Khan, M., Preventing Cavitation in Francis Runner, Water Power and Dam Cont., 1987, 15-21.
[8] Anup K.C., Young Hoo,L., Bhola, T., Cfd Study On Prediction Of Vortex Shedding In Draft Tube Of Francis Turbine And Vortex Control Technique, Renewable Energy, 86, 2016, 1406-1421.
[9] Ayancık, F., Çelebioğlu, K., Aradağ, S., Parametrical and Theoretical Design of a Francis Turbine Runner with the Help of Computational Fluid Dynamicsss, 10th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics, Orlando, Florida, USA, July 2014.
[10] Kelkar, A.S., A Revşew of Taguchi Techniques through Optimization of Cooling Rate of Turbine Blades, Institute of Technology, Nirma University, Ahmedabad, 382 481, December 2011.
[11] Versteeg, H.K., Malalasekera, W., An Intrıduction to Computational Fluid Dynamics The Finite Volume Method, 1st edition, 1995.