HEFAT2014
10th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
14 – 16 July 2014
Orlando, Florida

PARAMETRICAL AND THEORETICAL DESIGN OF A FRANCIS TURBINE RUNNER WITH THE HELP OF COMPUTATIONAL FLUID DYNAMICS

Ayancik, Fatma*, Celebioglu, Kutay and Aradag, Selin
*Author for correspondence
Department of Mechanical Engineering,
TOBB University of Economics and Technology,
Sogutozu Avenue 43, Sogutozu, Ankara, 06560,
Turkey
E-mail: fayancik@etu.edu.tr

ABSTRACT

A computational fluid dynamics based design system with the integration of blade modeler, mesh generator and Navier-Stokes based CFD codes makes the design optimization of turbine components quick and efficient. This design system is applied to a low head Francis turbine runner. The parameters of turbine runner affect the hydraulic performance of turbines. Its complex parameters cause direct effect on the global parameters which change the efficiency and the output power. The purpose of this study is the investigation of the effects of theoretical turbine runner parameters on the design. To determine the parameter effects on the turbine performance theoretical calculations and analyses of turbine runner were performed. A methodology was followed with the help of CFD to reach the best efficiency operating point of turbine. Starting from the preliminary design to the final design, theoretical calculations were performed and evaluated using the results of the CFD analyses. The CFD analyses were used to visualize the flow characteristics on runner blades induced by runner parameters. At the end, a new runner model is designed with a higher efficiency.

INTRODUCTION

The increasing efficiency improvement of hydraulic machinery and turbine performance has been an essential part for the industry of hydraulic machinery. To develop the performance and the efficiency, different design and manufacturing methods have been used since the radial flow type turbo machines which were invented by James Francis in 1855 [1]. From 1950's up to now, Computational Fluid Dynamics (CFD) has become one of the design methods with its trustable results. CFD makes internal flows in complex domains predictable and provides a straightforward way to visualize the main turbine parameters which have a big influence on the performances of hydro turbines [2,3].

To evaluate the design parameters, to prove the accuracy of the theoretical calculations and to explain the changes depending on the blade shape and turbine parameters; CFD has a part in the case analysed in this study beginning from the preliminary design to the final design. This paper presents the CFD-based design of a hydro turbine runner depending upon the parametrical and theoretical calculations to prevent cavitation and to increase the turbine efficiency. For one type of Francis turbine runner, all the parameters and their effects are illustrated with the help of CFD, considering the influence of the operating conditions, such as load, head and submergence [4]. Theoretical calculations and blade design characteristics are represented. The effects of several parameters on the design are also explained.

NOMENCLATURE

C	[m/s]	Absolute velocity
U	[m/s]	Peripheral velocity
W	[m/s]	Relative velocity
D	[m]	Runner diameter
H	[m]	Head
n_s	[-]	Specific speed
P	[kPa]	Pressure
Q	[m^3/s]	Discharge
g	[m/s^2]	Acceleration due to the gravity
b	[m]	Guide vane height

Special characters

α	[°]	Angle between C and U (Flow angle alpha)
β	[°]	Angle between W and U (Flow angle beta)
Γ	[-]	Circulation
η	[-]	Efficiency
σ	[-]	Thoma number
φ	[-]	Discharge coefficient
ψ	[-]	Energy coefficient
ω	[rad/s]	Angular velocity
ρ	[kg/m^3]	Density

Subscripts

1	High pressure side of machine
2	Low pressure side of machine
0	Guide vane outlet
d	Design
m	Meridional

CFD BASED DESIGN METHODOLOGY

The runner design process starts with the known parameters which are head and discharge determined by the topographical and hydrological features of the power plant [5]. Figure 1 shows the important angles for the runner blade.

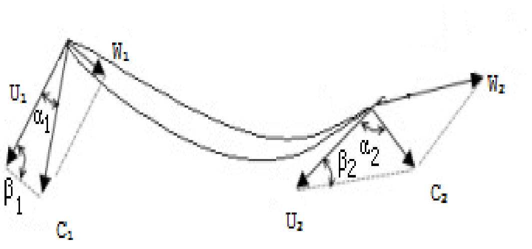

Figure 1 Blade flow angles (alpha, beta)

With these known values, all the initial design dimensions, specific speed and inlet flow angle are defined. To start the initial blade design, obtained values from preliminary design are converted to the beta angle and theta angle of the runner blade with the help of in-house Matlab codes. Afterwards, the blade design starts with coarse mesh and finishes when all the necessary conditions are obtained, as represented in Figure 2.

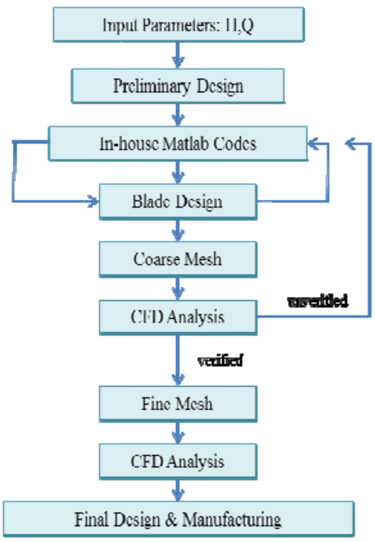

Figure 2 Design methodology [6]

Preliminary Design

The input parameters initialize the runner design procedure. First step of the preliminary design is the definition of the blade shape. The blade shape changes according to the specific speed of the turbine. The specific speed is one of the main parameters of all hydraulic turbines. It is used for the classification of the turbines by using the head, flow rate and speed as shown in equation (2). For Francis type hydraulic turbines, values of the specific speed changes between 60 and 330. Different values of the specific speed specify different initial blade shapes [7]. Low specific speed turbine blade shapes are designed radially to provide axial flow at the end of the runner exit. For high specific speed turbines, blade has short and high blades different from the low specific speed turbines as seen in Figure 3. To determine the specific speed, turbine design power is calculated as seen in equation (1).

Figure 3 Turbine runner blades at high (left) and low (right) specific speed

$$P_d = \rho g Q_d H_d \eta \quad (1)$$

Specific speed can be expressed using turbine design power:

$$n_s = n_{sync} \frac{P_d^{\,0.5}}{H_d^{\,1.25}} \quad (2)$$

Practically, with this value, general blade shape can be determined using the ranges of the specific speed. However; to calculate inlet and outlet diameters and flow angle alpha, equations seen below are used.

$$D_1 = 2U_1 / \omega \quad (3)$$

$$D_2 = \sqrt{(4Q_d)/(\pi C_{2m})} \quad (4)$$

$$\Gamma_0 = H_d \eta g 2\pi / \omega \quad (5)$$

$$\Gamma_0 = \pi D_{02} C_0 \cos\alpha_0 \quad (6)$$

The guide vane exit angle (flow angle) can be computed by using equation (5) and equation (6) together.

Beta and theta angles of the runner blade are calculated by using in-house Matlab codes and initial dimensions of the runner blade. These codes are generated by the theoretical design equations to find the optimum values of blade parameters. Matlab codes calculate the initial values of spiral casing and runner of the turbine by taking account of the energy equations and design equations. It is used for the iteration process.

Parameters and Parameter Effects

The turbine runner has a complex geometry which is affected by several design parameters.

The parameters shown in Table 1 are the main parameters which affect the turbine runner blade shape and efficiency. Beta angle is used to optimize the blade shape by generating linear or non-linear curves. This angle directly affects the rate of converted energy at each span of the runner from hydraulic to the mechanical. It also affects the velocity distribution on the blade surface [5]. Theta angle is directly dependent on the beta angle and its value changes with the integral of the beta angle. The other flow angle, alpha, gives a direction to the water and its direction causes more increment or decrement on the performance of the turbine according to the other parameters.

Table 1 Hydraulic design parameters

Parameters	Symbol	Unit
Specific speed	Ns	-
Flow angle alpha	α	degree
Rotational speed	N	rpm
Beta angle	β	degree
Theta angle	θ	degree
Guide vane height	b	m

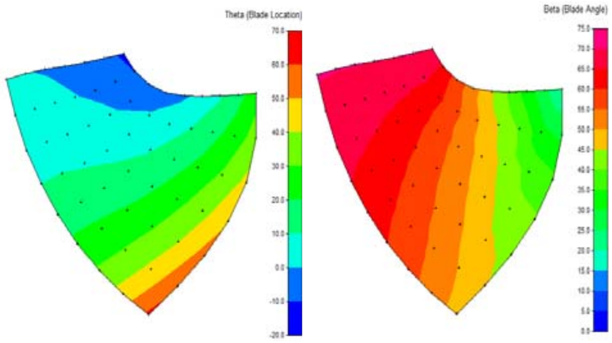

Figure 4 Theta and beta angles of the runner blade

To find the best values of the shape parameters like beta and theta angle, Bezier curves are preferred. These curves allow the improvement of a correlation between the control points and change the runner blade shape in a smooth way [2]. Cavitation depends on more than one parameter, for this reason; it can be named as a major parameter which decreases the efficiency and performance of the turbine if it is not fixed. It is affected by discharge, pressure and hydraulic design parameters indicated in Table 1. In addition to these, cavitation is related to the discharge and energy coefficient of the turbine because of their relation between each other indicated in equation (9) [4].

$$\varphi = \frac{Q}{\pi\omega\left(D_{ref}/2\right)^3} \qquad (7)$$

$$\psi = \frac{2E}{\omega^2\left(D_{ref}/2\right)^2} \qquad (8)$$

$$\psi = \psi\left(\varphi, \alpha_0\right) \qquad (9)$$

According to all these parameters, cavitation can be classified by their magnitude as seen below. It occurs where the local static pressure is equal or less than the vapour pressure and is stated by Thoma number [8]. Figure 5 shows cavitation-free blade pressure loading which corresponds to first condition ($\Delta\sigma > 0$).

$\Delta\sigma > 0$, without cavitation
$\Delta\sigma = 0$, cavitation inception
$\Delta\sigma < 0$, cavitation
$\Delta\sigma << 0$, super-cavitation

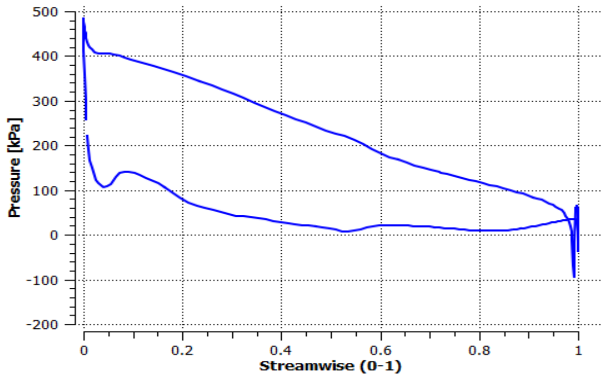

Figure 5 Cavitation-free blade loading

Quick Analysis

At the beginning of the design process, the runner blade is created with the help of BladeGen. Its flow direction, rotation direction and initial blade angles obtained from Matlab codes are defined. Then, the iteration process starts by combining in-house codes and CFX. The process continues until the area of the blade remains unchanged. Matlab codes check every span on the blade surface to adjust the blade angles correctly. Afterwards, with the defined profile and the appropriate angles, preliminary design is completed. In this case, to find the best starting points, firstly; coarse mesh is preferred. After a few iterations when the CFD results become closer to the requested parameter values, fine mesh analyses are performed.

COMPUTATIONAL MODEL BASED ON CFD

To determine the flow characteristics on the blade surface and the other turbine components like spiral casing, stay vanes, wicket gates and draft tube, CFD analyses are performed. CFD setup is prepared with given parameters. Boundary conditions are used as an initial value to start analyses. The runner inlet angle is defined as an initial condition for the other components of the turbine. Design starts from the runner. Its initial angle and dimensions are taken as reference for the other components of the turbine. After runner design, guide vane, stay vane and spiral casing are designed consequently. Draft tube is designed according to the runner outlet angle since runner outlet swirl is an important point which should be considered for draft tube design [9]. In this case, as mentioned in literature, 15 runner blades, 24 guide vanes and 16 stay vanes are used [10]. Figure 6 indicates the position of the components of the turbine.

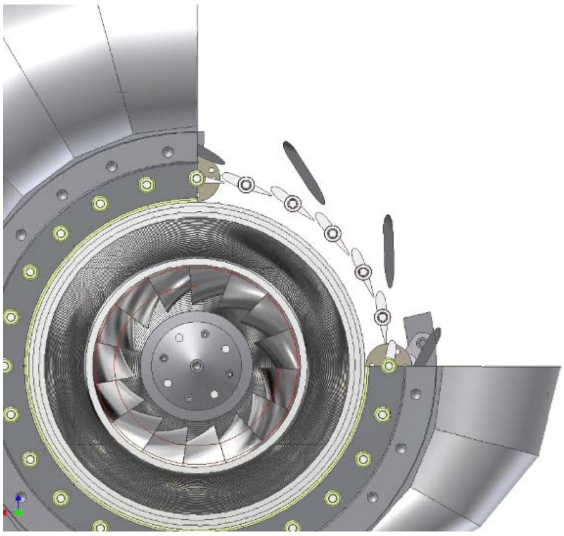

Figure 6 Francis type turbine components cross-section view

Table 2 Inlet conditions of the turbine components

Component	Inlet angle	Outlet angle	Unit
Stay vane	37.1	37.1	degree
Guide vane	37.1	29.6	degree
Runner	29.6	5.4	degree

Boundary conditions

Boundary conditions are defined as pressure inlet and mass flow outlet. k-ε turbulence model is chosen due to its robustness in solving steady state turbulent flows [11]. Turbulent flow inside is solved by using Reynolds Average Navier-Stokes equations. In this equation, all the variables are time averaged and all the instantaneous variables are decomposed in mean and fluctuating values by Reynolds decomposition [6].

$$\rho\frac{\partial}{\partial t}\left(u_j\right)+\rho u_k\frac{\partial}{\partial x_k}\left(u_j\right)=-\frac{\partial P}{\partial x_j}+\mu\frac{\partial^2}{\partial x_i \partial x_j}\left(u_j\right)+\rho f_j \qquad (10)$$

Mesh generation

Mesh generation is another step in the CFD based design process. It is important to obtain accurate results. To determine the initial shape of the runner blade, coarse mesh is preferred due to its easiness and quickness. In the main design process, fine mesh is applied. H-J-C-L grid mesh is preferred. Runner blade is discretized by the combinations of the more than one topology to grab the changeable velocity gradients accurately [11]. Close vicinity of the runner blade is meshed with clustering O-grid. H and L grid topology are applied in upstream and downstream regions of the blade.

RESULTS

CFD based design process starts with preliminary design and given parameters. Afterwards, iterative processes take part in the design study with the defined boundary conditions. Initial blade shape and the final blade shape design can be seen in Figure 7.

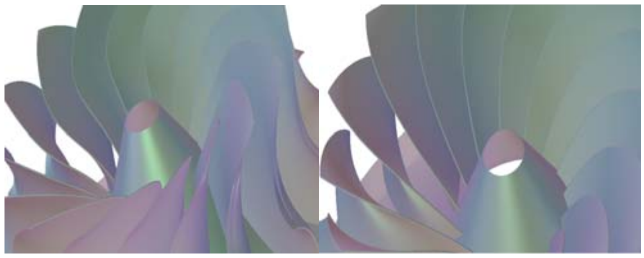

Figure 7 First (left) and final (right) runner blade design

Blade shape design is dependent upon the parameters and CFD results. Given parameters provide information about how the results must be. Velocity, pressure, head, power and efficiency are the main criteria to compare the results with the requested values. Flow angles are the second set of parameters to check. Velocity distribution and velocity vectors on the blade surface give more details about the pressure profile and flow distribution. Pressure and head give detail about cavitation and pressure distribution on the blade surface. Flow angle alpha provide prediction about the draft tube inlet and the runner outlet.

According to data and results obtained from similar process, Drtina and Sallaberger found that absolute and relative velocity components vary in the circumferential direction of the flow. This causes and also affects the 3D character of the flow. At the operating condition of the turbine, streamlines on the blade surface should be parallel to the leading edge as indicated in left side of Figure 8 [10]. The right side shows the final design of the runner blade in this case.

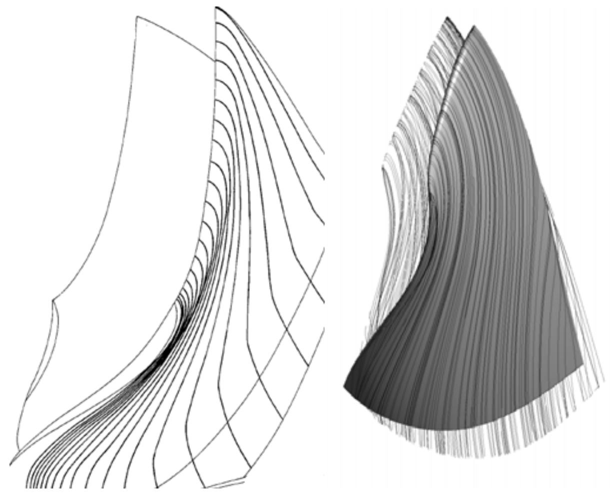

Figure 8 Streamlines on the final design blade surface [10]

As seen in Figure 8, optimized runner blade shape is smooth and good enough to show the best 3D flow character at the optimum discharge value. Same as the streamlines, velocity vector distribution must show the smooth manner on the blade surface as seen in Figure 9. All the flow velocity vectors must follow the surface without any separation [12].

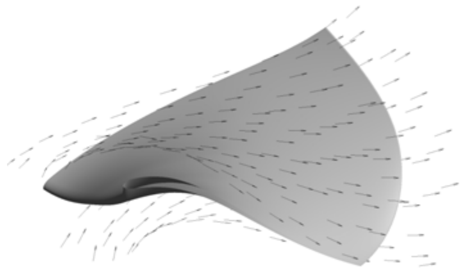

Figure 9 Velocity vectors on the runner blade surface

To obtain higher efficiency and to provide enough power, necessary conditions are uniform pressure distribution on the blade surface as shown in Figure 10 and Figure 11 and smooth velocity vector distribution as shown in Figure 9. Mesh quality around the blade, turbulence model, inlet and outlet angles and hydraulic losses are also the other conditions which affect the efficiency. On the other hand, some results might be deceptive while looking at the result table. Therefore, some calculations should be made to check the accuracy of the analyses and to prevent the errors for turbine runner.

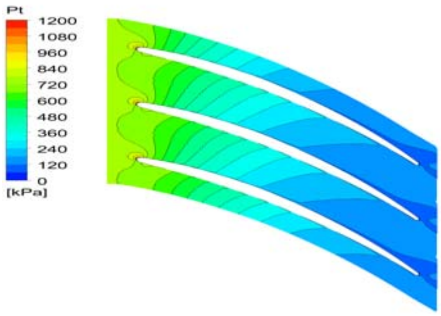

Figure 10 Total pressure distribution at the blade to blade view

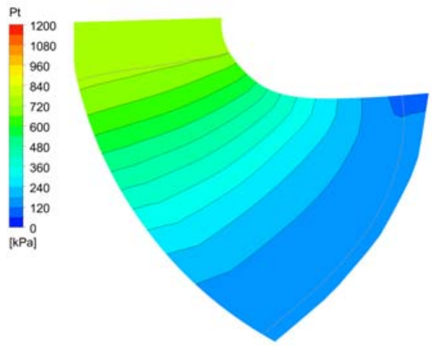

Figure 11 Total pressure distribution at the meridional view

Equations are based on Bernoulli and Euler Energy equations to compare the results [13].

$$H\eta_h = \frac{1}{g}(U_1 C_1 \cos\alpha_1 - U_2 C_2 \cos\alpha_2) \qquad (11)$$

$$H\eta_h = \frac{\omega}{g 2\pi}\left(\Gamma_1 - \Gamma_2\right) \qquad (12)$$

$$H\eta_h = \frac{C_1^2 - C_2^2}{2g} + \frac{U_1^2 - U_2^2}{2g} + \frac{W_2^2 - W_1^2}{2g} \qquad (13)$$

The left side of the equations is the energy expressed in J received by the runner from the liquid weighing 1N, passing through the runner blades. Equation (12) contains the kinematic parameters of the flow upstream and downstream of the runner [13]. Starting from the equation (13) to the equation (11), calculations are completed by using the previous result of the equations. The main objective of the calculations is to find α (flow angle alpha) to compare it with the given parameters and then to check the circulation found in equation (12). Outlet circulation must be close to zero to prevent cavitation [13].

To have a complete understanding and to increase the clarity of the analyses, CFD based design process is conducted with the other turbine components. Their initial conditions are obtained from the results of the runner blades. Any alteration on the runner blade affects the components of the turbine. Figure 12, 13 and 14 show the results of the other turbine components.

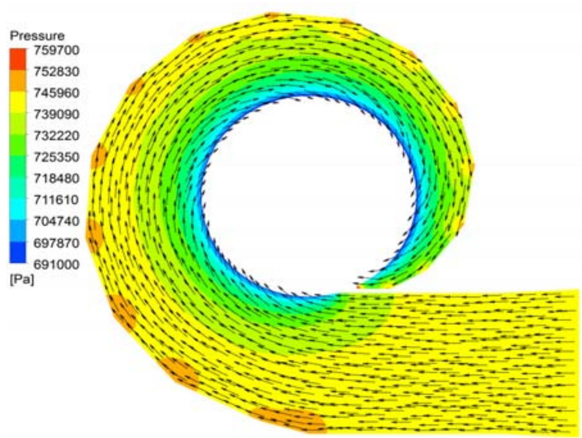

Figure 12 Pressure contour and velocity vector distribution on the mid-plane of the spiral casing [9]

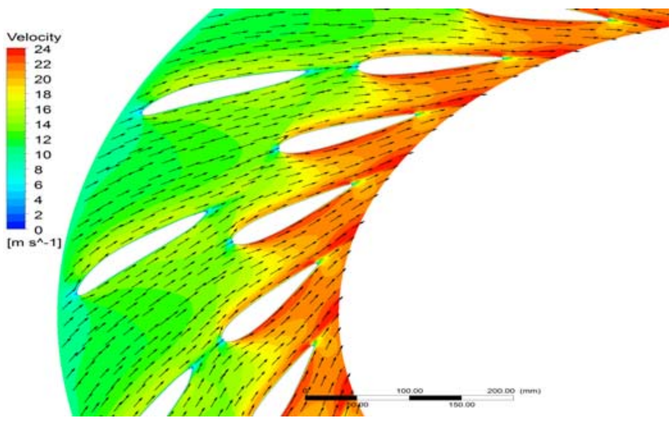

Figure 13 Velocity contour and velocity vector distribution on the tandem-cascade [9]

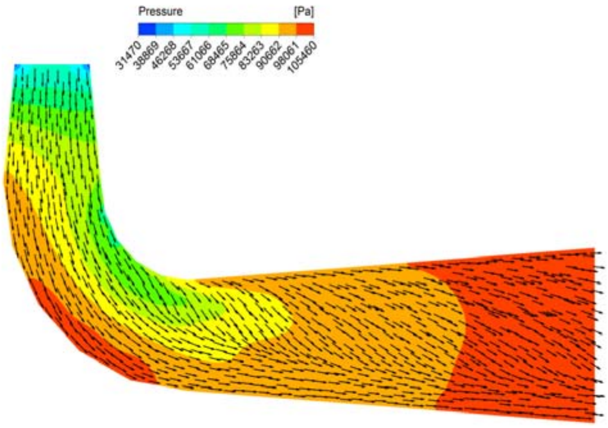

Figure 14 Pressure contour and velocity vector distribution on the mid-plane of the draft tube [9]

DISCUSSIONS AND CONCLUSION

Parametric design based on numerical calculations indicates that, hydraulic design parameters cause different effects on the blade shape and the turbine performance. Parametric relations between the beta and theta angle show that the beta angle distribution has an important role in the design process. Cavitation parameters and cavitation effects change according to the beta distribution. It is found that for different beta and theta configurations of a single turbine runner, performance and cavitation are affected.

In this paper, five parameters and their effects on the final design are investigated. To define the parameters five different non-linear Bezier curves are used. The influences of these parameters are determined with CFD results. According to the results, final blade shape is determined by changing only flow angle alpha and blade angles.

Mesh study is performed depending on the results. Coarse mesh results gave quick information about the performance. Fine mesh and high resolution advection scheme gave the required and more accurate results about performance and efficiency.

To compare the final theoretical and numerical results, previous studies are used. All the result provided the required conditions and the design finalized.

ACKNOWLEDGEMENT

This study is financially supported by the Ministry of Development of Turkey. The computations are performed at TOBB ETU Center for Hydro Energy Research (ETU Hydro), CFD Laboratory.

REFERENCES

[1] Round G.F., Incompressible flow Turbomachines: Design, selection, applications and theory, *USA, Gulf Professional Publishing*, 2004, pp. 8-24

[2] Grapsas V.A., Anagnostopoulos J.S., and Papantonis D.E., Hydrodynamic design of radial flow pump impeller by surface parameterization, *Proceedings of the 1st International Conference on Experiments/Process/System Modelling/Simulation/Optimization (IC-EpsMsO),* Athens, Greece, July 2005.

[3] Wang L., The optimal design based on CFD combined with CAD for turbine runner, *Journal of Software,* Vol.7, No. 8, August 2012, pp.1721-1726

[4] Avellan F., Introduction to cavitation in hydraulic machinery, *Proceedings of the 6th International Conference on Hydraulic Machinery and Hydrodyamics,* Timisora, Romania, October 2004, pp. 11-22

[5] Thapa B.S., Thapa B., Eltvik M., Gjosater K., and Dahlhaug O.G., Optimizing runner blade profile of Francis turbine to minimize sediment erosion, *Proceedings of the 26th IAHR Symposium on Hydraulic Machinery and Systems,* Beijing, China, Paper number 032052, 19-23 August, 2012.

[6] Okyay, G. Utilization of CFD tools in the design process of a Francis turbine, *M.S. thesis, Department of Civil Engineering Middle East Technical University*, Ankara, Turkey, 2010.

[7] Raabe J., Hydropower: the design, use, and function of hydromechanical, hydraulic and electrical equipment, VDI-Verlag, Verlag des Vereins Deutscher Ingenieure, Düsseldorf, 1985

[8] Muntean S. Câmpian V.C., Cuzmoş A., Dumbrava C., Brebu N., and Augustinov L., Experimental ınvestigations into a Francis turbine with low specific speed, *U.P.B. Scientific Bulletin Series D; Mechanical Engineering,* Vol. 73, 2009

[9]Akin H., Celebioglu K., Aradag S., A CFD based design methodology for hydraulic turbines applied to a case study in Turkey, Accepted for publication in the *10th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics,* July 2014.

[10] Drtina P., Sallaberger M., Hydrulic turbines basic principles and state of the art computational fluid dynamics applications, *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science,* Vol. 213, July 1998.

[11] Wu J., Shimmei K., Tani K., Niikura K., Sato J., CFD based design optimization for hydro turbines, *Journal of Fluids Engineering,* Vol. 129, February 2007, pp.159-168

[12] Ayancik F., Aradag U., Ozkaya E., Celebioglu K., Unver O., and Aradag S., Hydroturbine runner design and manufacturing, *International Journal of Materials, Mechanics and Manufacturing,* Vol. 1, No.2, 2013, pp.162-165

[13] Krivchenko G.I., Hydarulic machines: Turbines and pumps, *Moscow, Mir Publishers*, 1986, pp.73-77