A CFD-BASED DESIGN METHODOLOGY FOR HYDRAULIC TURBINES APPLIED TO A CASE STUDY IN TURKEY

Akin H.*, Celebioglu K. and Aradag S.
*Author for correspondence
Department of Mechanical Engineering,
TOBB University of Economics and Technology,
Ankara, 06560,
Turkey,
E-mail: hakin@etu.edu.tr

ABSTRACT

Hydraulic turbines are turbo machines which produce electricity from hydraulic energy. Francis type turbines are the most common one in use today. The design of these turbines requires high engineering effort since each turbine is tailor made due to different head and discharge values. Therefore each component of the turbine is designed specifically. During the last decades, Computational Fluid Dynamics (CFD) has become very useful tool to predict hydraulic machinery performance and save time and money for designers. This paper describes a design methodology to optimize a Francis turbine by integrating theoretical and experimental fundamentals of hydraulic machines and commercial CFD codes.

INTRODUCTION

Hydropower, the largest source of renewable energy, is a clean and very efficient way to generate electricity in Turkey and many countries around the world. The machines which produce electricity from hydraulic energy are the hydraulic turbines. Francis turbines are the most popular one among all other hydraulic turbines due to their vast operating regime and high efficiencies up to 95% [1]. Each turbine is unique to the project because of different head and discharge, therefore each component of the turbine of a power plant must be designed specifically. The traditional design is carried out by means of experiments, measurements and model tests, which require significant money and time. With increasing computational power during the last two decades, Computational Fluid Dynamics (CFD) has become an important tool for the design of turbo machines. Complex, turbulent, three-dimensional flows occurring in the entire turbine can be solved by CFD which is a cheap and fast way to predict turbine performance other than model tests. With the help of CFD, modifications in the design can be implemented by analyzing the flow pattern inside turbine components in detail. Turbine efficiency and power can be increased and undesired situations such as not matching flow angle, vortex or cavitation in the fluid domain can be avoided by making necessary modifications in the components. However, in order to check the reliability of the optimized turbine, the results should be validated with model tests. CFD has been an additional element in the design process which saves money and time [2-4].

NOMENCLATURE

H	[m]	Net available head
Q	[m³/s]	Net available flow rate
n	[rpm]	Rotational speed
D	[m]	Diameter
L	[m]	Length
R	[m]	Radius
V	[m/s]	Velocity
K	[m²/s]	The factor of velocity moment
b	[m]	Width
g	[m/s²]	Gravity
h	[m]	Head
c	[-]	Recovery factor

Special characters

α	[°]	Flow angle
θ	[°]	Cone angle
φ	[°]	Coverage angle

Subscripts

g	Guide vane
s	Stay vane
r	Radial direction
u	Circumferential direction
st,out	Stay ring outlet
0	At the wicket gate
1	Inlet
2	Outlet
c	Cone
d	Diffuser
l	Loss
p	pressure

Francis turbine, a reaction turbine, may be divided in two groups; the one group with horizontal and the other group with vertical shaft. While some small turbines are arranged with horizontal shaft, larger turbines are arranged with vertical shaft [5]. The flow domain of a Francis turbine with vertical shaft is shown in Figure 1, comprising spiral case, stay vanes, guide vanes, runner and draft tube.
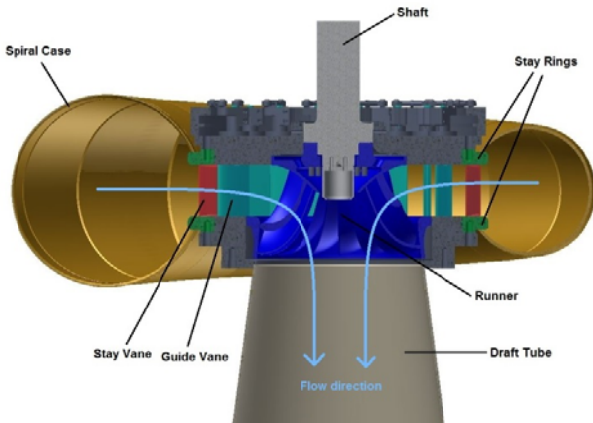

Figure 1 Half-section view of a Francis turbine

Water enters the spiral case which surrounds the runner. The cross-sectional area of the spiral case decreases along the flow path in order to provide uniform velocity distribution in radial and circumferential directions at the inlet to stay vanes. Stay vanes are responsible for carrying pressure loads in the spiral case and cover and directing the flow towards guide vanes with an appropriate incidence angle. The flow rate and thus the power of the turbine are controlled by guide vanes. Guide vanes also provide an optimum flow angle at the inlet of the runner. The flow in the runner is radial at the inlet and axial at the outlet. The angular momentum of the water is reduced and work is supplied to the shaft through the runner. Water is discharged from runner outlet to tailwater by a diffuser called draft tube. The cross-sectional area of the draft tube increases along the flow path to recover the maximum energy [6,7].

In the present paper, a design methodology is described to design a Francis turbine by integrating theoretical and experimental fundamentals of hydraulic machines and commercial CFD codes. The methodology is applied to an actual Francis turbine which will be installed in Turkey. Necessary input variables are the available net head: 66.8 m, and the available volume flow rate: 4.25 m³/s.

DESIGN METHODOLOGY

Francis turbines are composed of five components considered in flow analysis, spiral case, stay vanes, guide vanes, runner and draft tube. All components have many variables affecting the turbine characteristics and they are also dependent on head and discharge. Therefore, a parametric design procedure is needed to design the components of a unique Francis turbine. The design methodology followed in this study is outlined in the flow chart shown in Figure 2. The methodology consists of three main steps: preliminary design,

geometry generation and CFD. It is repeated until a fully optimized model with satisfactory performance is obtained.
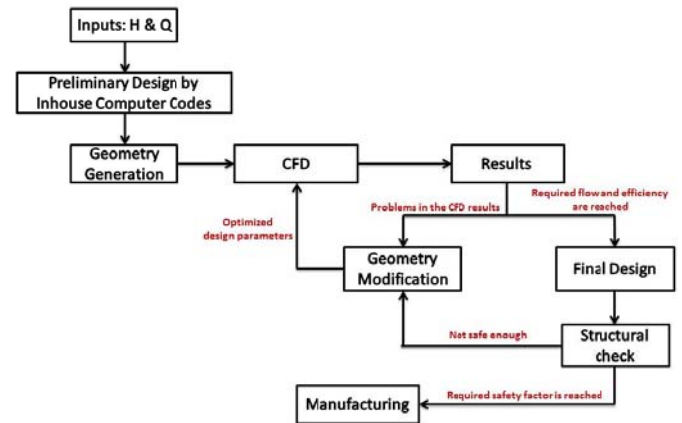

Figure 2 Design methodology

Initial dimensions of the components are determined by using experimental and theoretical data in the state-of-the-art of hydraulic turbines at the preliminary design stage. The preliminary design starts with runner. The specific speed of the turbine is calculated with given head (H) and discharge (Q) values and determined rotational speed, n (rpm). Empirical values for a Francis turbine runner and the meridional profile as a function of the specific speed are obtained [8]. NACA profiles are defined for meridional sections. Flow conditions and blade angles at the inlet and the outlet of runner are provided by Euler equations. Distribution of blade angles for meridional sections is optimized with CFD results iteratively.

After the dimensions of runner are settled, the guide vane is designed. The width of the guide vane is determined by the width of the runner inlet. The diameter of the circle where guide vanes rotate about their axis, D_g as shown in Figure 3, is generally chosen as 1.16 of the runner inlet diameter [9]. The number of guide vanes is generally determined as 12, 16 or 24. The guide vane length, L_g, with 10% overlap at closed position can be calculated as follows [9]:

$$L_g = \pi D_g / (0.9 * \text{number of guide vanes}) \qquad (1)$$

The rotation angle of guide vane at the optimum opening case, α_g, is determined in a way to provide the correct inflow angle at the runner inlet by CFD analysis.
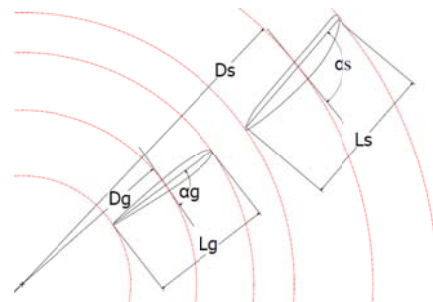

Figure 3 Tandem cascade

Once the dimensions of the guide vane are settled, the diameter of the circle where stay vanes are assumed to rotate about their axis, D_s as shown in Figure 3, is determined by experience. The main function of stay vanes is structural and the cross sectional area of the blade is important. Therefore, the length of stay vane, L_s, and thickness of the blade are chosen in a way to carry the forces acting on themselves. While symmetric NACA profiles are preferred at guide vane design, asymmetric profiles determines the thickness of the stay vane. Besides, blade rotation angle, α_s, and blade profile are optimized in order to provide accurate incidence angle to guide vanes during CFD analysis.

The aim of spiral case is to distribute the flow uniformly around the stay vanes. This uniform flow distribution can be provided by using the theoretical "the law of constancy of the flow velocity moment" through spiral case. The flow angle at the outlet of the stay ring, $\alpha_{st,out}$, preserves a constant value over the entire perimeter of the wicket gate with a constant K. Spiral case design is practiced at a constant $\alpha_{st,out}$ coinciding with the angle of guide vanes [9]. The diameters and the distances of spirals to the center are calculated in a way to obtain equal K values at the sections divided by coverage angle, φ_{cov}. The angle $\alpha_{st,out}$ depicted in Figure 4 can be expressed as given in equation (2) .
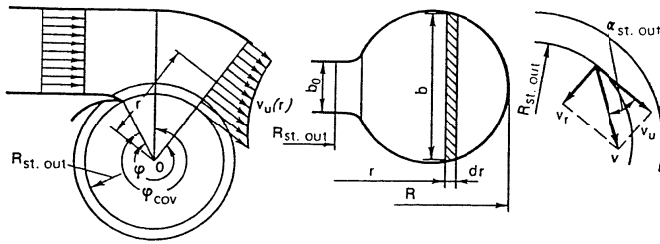

Figure 4 Calculating the spiral case [9]

$$\tan \alpha_{st,out} = \frac{v_r}{v_u} = \frac{QR_{st,out}}{\pi D_{st,out} b_0 K} = \int_{R_{st,out}}^{R} \frac{b(r)}{r} dr / (2\pi b_0 \frac{\varphi}{360}) \quad (2)$$

An elbow-type draft tube as presented in Figure 5 is used to discharge the water from the runner outlet through tailwater. The dimensions of the draft tube are based on the outlet diameter of runner, which is equal to inlet diameter of draft tube D_1. The height of cone L_c, the cone angle θ_c, the length of the diffuser L_d, the expansion angle of diffuser θ_d are important parameters affecting the performance of the draft tube. Initial values of the parameters are obtained from experimental data [9, 10] optimized with CFD.

While optimizing the parameters of draft tube, draft tube efficiency is investigated. Draft tube efficiency is described by pressure recovery factor given in equation (3). For well-designed draft tubes, pressure recovery factor, c_p, reaches 80-85% [10].

$$c_p = (\frac{V_1^2}{2g} - \frac{V_2^2}{2g} - h_L) / (\frac{V_1^2}{2g}) \quad (3)$$
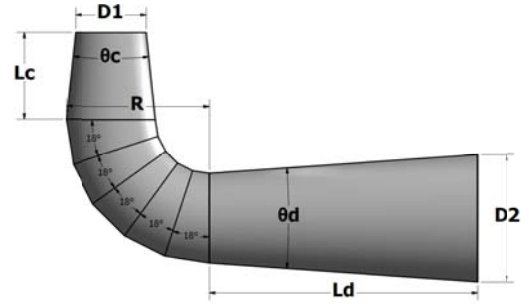

Figure 5 Draft tube

At the geometry generation stage, the flow domains of the components the initial dimensions of which have been determined are created for CFD simulations. All blade profiles including stay vanes, guide vanes and runner are generated using *ANSYS BladeGen* tool [11]. The flow domain geometries of spiral case and draft tube are modeled by using a 3D CAD program, *Autodesk Inventor* [12]. When all geometries are ready, CFD process begins.

CFD METHOD AND COMPUTATIONAL DETAILS

A commercial RANS code, *ANSYS CFX 14.0* [11], is used for the turbulent flow simulation to obtain physically accurate solutions. The main components, spiral case, stay vanes, guide vanes and draft tube, are considered in flow simulation separately due to the complexity of the flow inside whole Francis turbine and computational effort. In the final optimization, the flow through spiral case, stay vanes and guide vanes are simulated together to validate the results of the single analyses.

The computational domains of the spiral case and the draft tube are meshed using unstructured tetrahedral/hexahedral elements in *ANSYS ICEM CFD*. On the other hand, stay vane and guide vane geometries, which are transferred from *ANSYS BladeGen* into *ANSYS Turbogrid*, are meshed using the available topology sets like H/J/C/L grids & O grids. The summary of generated mesh for each component is presented in Table 1 and mesh profiles of the spiral case with tandem cascade and the draft tube are given in Figure 6 and Figure 7.
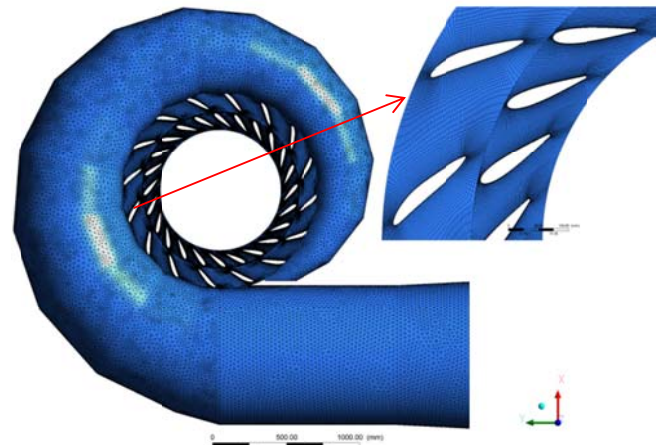

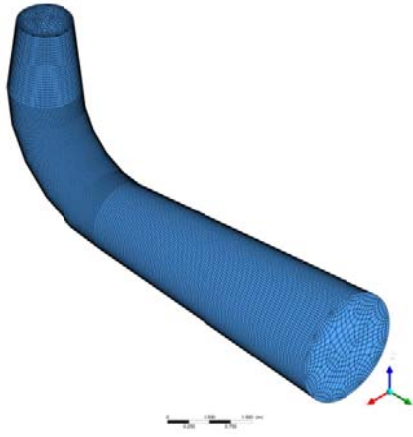**Figure 6** Computational grid of the spiral case with tandem cascade

Figure 7 Computational grid of the draft tube

Table 1 Summary of mesh data

Domain	Number of Nodes	Number of Elements	Type of Element
Spiral Case	166640	869250	Tetrahedral
Stay Vanes	252188	237072	H/J/C/L & O Grids
Guide Vanes	253080	237720	H/J/C/L & O Grids
Draft tube	70686	67064	Hexahedral

The boundary conditions are specified as total-pressure inlet and mass flow outlet for all components except the draft tube. For the draft tube simulations, boundary conditions are mass flow inlet, which is in the direction of runner outlet, and static pressure outlet. 757025.4 Pa (66.8 m+10.3 m) total pressure value is given as inlet condition and 4250 kg/s mass flow value per machine is given as outlet condition at the simulations for spiral case and tandem cascade. 4250 kg/s mass flow inlet and 1 atm static pressure outlet are boundary conditions values for the draft tube simulations. Besides, the reference pressure is set to 0 atm. Simulations are performed as steady state. Rapid simulations are performed to determine the overall geometry of components using coarser mesh and upwind advection scheme. For the final simulations, fine mesh and high resolution advection scheme are used. k-ε turbulence model is used due to its robustness in practical applications [14] and the walls of all domains are assumed to be smooth with no slip. The convergence criteria for the RMS residuals of pressure and velocity is 10^{-5} in all simulations.

RESULTS

The important CFD results such as pressure distribution, velocity distribution, efficiency, head losses and flow angles are investigated after the simulations are completed. Incorrect flow angles, uneven pressure distribution and flow separation are easily detected viewing related flow contours. The results of the final design of spiral case, tandem cascade and draft tube are presented below.

The equal distribution of the water around the runner is significant for a balanced operation of the turbine. Velocity

distribution at the outlet of the spiral case is plotted in Figure 8. It is seen that the spiral case is adequate to provide a uniform distribution of radial and the circumferential velocity at the inlet of tandem cascade.
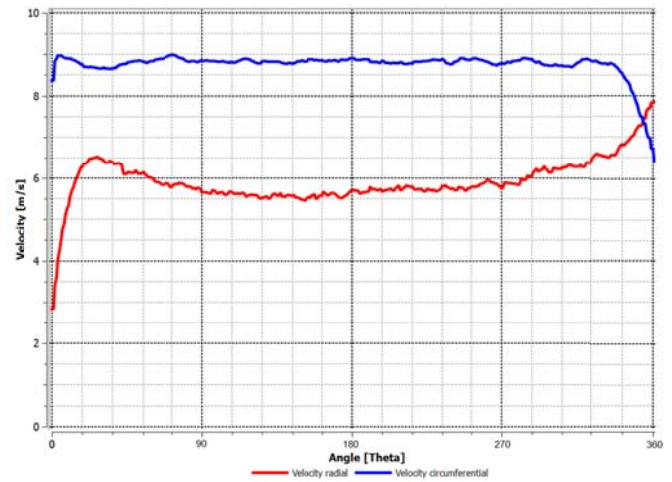

Figure 8 Velocity distributions at the outlet of the spiral case

The pressure distribution on the mid-plane of the spiral case is displayed in Figure 9. Pressure decreases gradually through the outlet of spiral case, since flow accelerates. An equal pressure around the outlet also indicates a uniform flow at the inlet of tandem cascade. Besides velocity vectors shows a good flow distribution through tandem cascade.
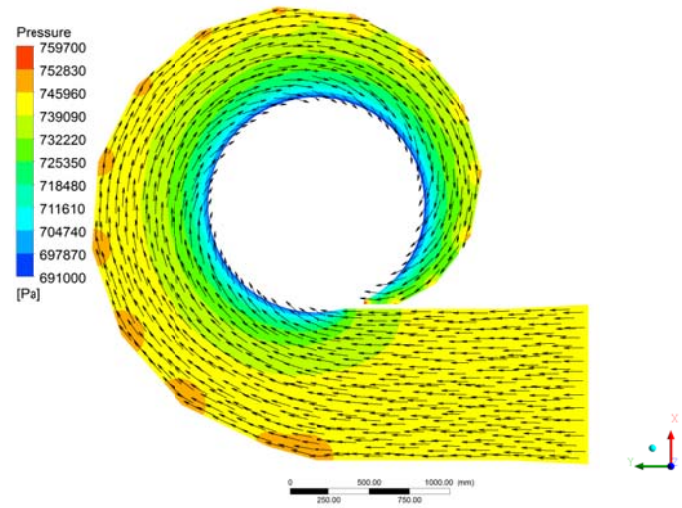

Figure 9 Pressure contour and velocity vectors on the mid-plane of the spiral case

The pressure distribution on the mid-plane of tandem cascade is shown in Figure 10. Stay vanes are desired to be fore-loaded. As it can be seen from this figure, the stagnation point caused by stay vanes is slightly behind of the symmetry point at the leading edge of stay vanes, which shows that the outflow angle of spiral case matches well with the leading edge of stay vanes. On the other hand, the stagnation point caused by

guide vanes coincides quite well with the symmetry point of leading edge of guide vanes as expected. This indicates that stay vanes deliver the flow through guide vanes with the correct flow angle.
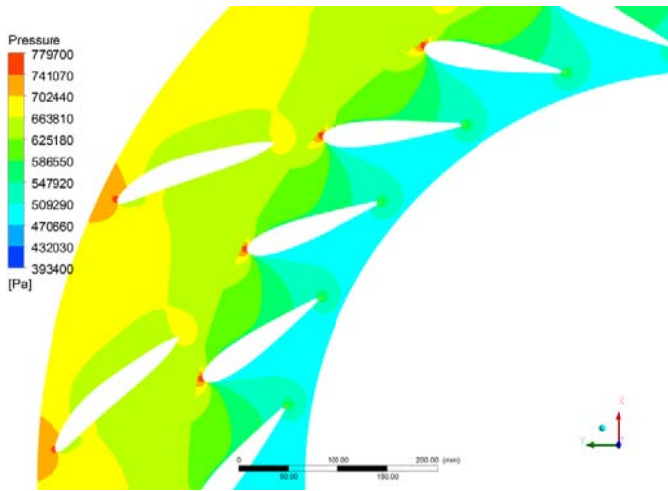

Figure 10 Pressure contour on the mid-plane of tandem cascade

The velocity distribution on the mid-plane of tandem cascade is displayed in Figure 11. The flow is guided with correct angle and minimum hydraulic loss to the runner. Any backflow or flow separation is not observed in flow area, as shown in figure.
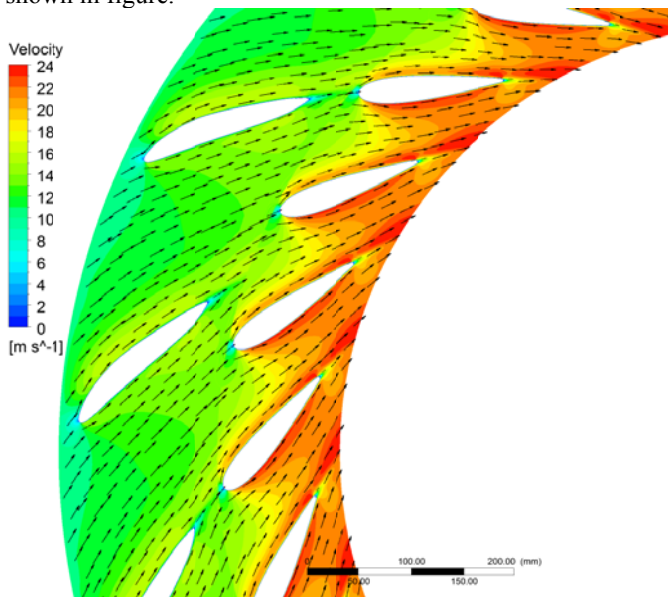

Figure 11 Velocity contours and vectors on the mid-plane of tandem cascade

Figure 12 shows the velocity vectors and the pressure distribution on the mid-plane of draft tube. The flow moves in a good manner from the inlet of the draft tube through the outlet of the draft tube without any flow separation. Static pressure increases in the flow direction, which is the main role of the draft tube. Draft tube operates with the pressure recovery factor of 0.825.
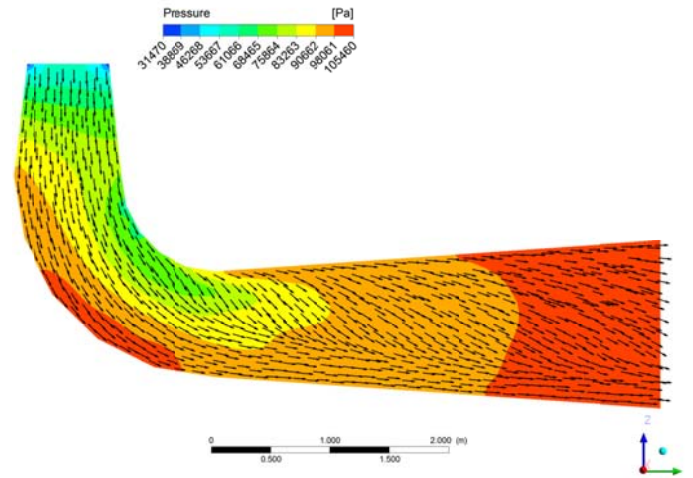

Figure 12 Pressure contours and velocity vectors on the mid-plane of the draft tube

The cumulative head loss of spiral case, tandem cascade and draft tube is calculated as 3.2 m. As a result, runner has an available net head of 63.6 m. Cavitation-free runner blades are designed with hydraulic efficiency of 97.8% [13]. The overall efficiency of the final design is 92.4% and the shaft power of the turbine is 1.15 MW.

CONCLUSION

A CFD-based design methodology, which integrates theoretical and experimental fundamentals of hydraulic machines and commercial CFD tools, is presented in this paper. It allows a quick and efficient development and optimization of turbine components. The methodology is applied to the design of a Francis turbine. The results of the final design satisfy all hydraulic and desired performance requirements. The coupled simulations guarantee a flow match between different components. In conclusion, it is seen that the methodology is found to be working properly. The turbine the design of which has been completed is at the stage of manufacturing and will be ready to generate electricity in a short time.

ACKNOWLEDGMENT

REFERENCES

[1] Bobar M., Zsembinszki S., Muntean S., Nedelcu D., Hydrodynamic design and numerical analysis of a new Francis turbine with high specific speed, Proceedings of the *International Conference on Hydraulic Machinery and Equipments*, Romania October 2008

[2] Lain S., Garcia M., Quintero B., Orrego S., CFD numerical simulations of Francis turbines, Proceedings of the Revista Facultad de Ingeniería Universidad de Antioquia, pp. 24-33, February, 2010

[3]   Shukla M. K., Jain R., Prasad V., Shukla S. N., CFD analysis of 3-D flow for Francis turbine, MIT International Journal of Mechanical Engineering ,Vol.1, No.2 pp.93-100, 2011.

[4]   Keck H., Sick M., Thirty years of numerical flow simulation in hydraulic turbomachines, Proceeding of the. Acta Mechanica, 201, pp.21-229, 2009

[5]   Kjolle A., Hydropower in Norway: Mechanical equipment, Trondheim, December, 2001

[6]   Drtina P., Sallaberger M., Hydraulic turbines-basic principles and state-of-the art computational fluid dynamics applications, Proceedings of the Institution of Mechanical Engineers, 213(C), 1999, pp.85-102

[7]   Munson, Young, Okiishi and Huebsch, Fundamentals of fluid mechanics, 6th ed. Asia: Wiley, 2010, ch.12.

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

[9]   Krivchenko G.I., Hydraulic machines: turbines and pumps, Moscow: Mir Publishers, 1986

[10] Gubin M. F., Draft tubes of hydro-electric stations, Energiya Press, Moscow, 1970

[11] ANSYS 14.0 Workbench User Guide

[12] Autodesk Inventor 2014 Help

[13] Wu, et al., CFD-based design optimization for hydro turbines, ASME Journal of Fluids Engineering, 129, 2007, pp. 159-168

[14] Ayancik F., Celebioglu K., Aradag S., Parametrical and theoretical design of a Francis turbine runner with the help of computational fluid dynamics, Accepted for publication in the 10th International Conference on Heat Transfer, Fluids Mechanics and Thermodynamics, Florida, July, 2014