Hydroturbine Runner Design and Manufacturing

Fatma Ayancik, Umut Aradag, Ece Ozkaya, Kutay Celebioglu, Ozgur Unver, and Selin Aradag

***Abstract*—This research describes a methodology for the parametric design, computational fluid dynamics (CFD) aided analysis and manufacturing of a Francis type hydro turbine runner. A Francis type hydro turbine consists of five components which are volute, stay vanes, guide vanes, runner and draft tube. The hydraulic performance of the turbine depends on the shape of the components; especially on the shape of the runner blades. The design parameters for the other components are affected by the runner parameters directly. Runner geometry is more complex than the other parts of the turbine. Therefore; to obtain accurate results and meet hydraulic expectations, CFD analyses and advanced manufacturing tools are necessary for the design and manufacturing of the hydro turbine runner. The turbine runner design methodology developed is presented using an actual potential hydraulic power plant in Turkey.**

***Index Terms*—CFD, francis turbine, runner, design and manufacturing.**

I. Introduction

Turbines are used for hydropower generation. There are basically two types of hydraulic turbines, the first one is impulse and the second one is reaction type turbines. Impulse turbines work based on momentum principle; while in the reaction type turbines, the flow is fully pressurized and it works according to conservation of angular momentum [1].The potential energy of fluid is converted to kinetic energy. Francis and Kaplan type turbines are examples of reaction turbines [2]. Francis type turbines have a wide range of specific speed. Furthermore; these are the most commonly used hydraulic turbines for hydropower generation.

Francis type turbines are composed of five components. These are volute, stationary vanes, guide vanes, runner and draft tube [3]. Volute is designed to keep the velocity distribution uniform in the circumferential direction and it also converts pressure head into velocity head. Stationary vanes carry pressure loads in the volute and they provide the flow to reach the guide vanes without hydraulic losses. Guide vanes are the movable components of a Francis turbine. These are connected to the shafts to provide appropriate design angles to the runner inlet and also to control the flow, thus the power output of the turbine. The main component of Francis turbines is the runner. The runner decreases the pressure and angular momentum of the fluid and this imparts reaction on the runner blades and as a result, power is generated [4]. The last component of Francis turbines is the draft tube. It connects the runner and the tailwater. Because of its shape, water pressure increases along the tube which provides maximum pressure recovery [3].

Design and optimization of these components is crucial. Especially, runner design affects the parameters for all other turbine components. For this reason; design of the runner should provide most of the requirements and constraints. High level of efficiency and cavitation free flow on the runner blades is the necessary requirements according to Daneshkah, K. and Zangeneh, M. [5]. Runner geometry is complex and rotational; therefore to get accurate results, CFD (Computational Fluid Dynamics) is widely used. CFD tools help to determine the flow characteristics throughout the runner.

In this study, the design and manufacturing methodology for the runner of the turbines of hydraulic power plants is explained. The aim of this study is to express this design and manufacturing methodology for hydro turbine runners with the help of a case study: Turbine runner of Yuvacik Hydro-Electric Power Plant (H.E.P.P.) in Turkey.

Currently, a center for the design, manufacturing and tests of hydro-turbines, is under construction at TOBB University of Economics and Technology, as the output of a project granted by Turkish Ministry of Development. This paper describes the methodology developed as a part of this project for the design and manufacturing of the most important part of hydro-turbines: turbine runner, based on the case study of Yuvacik HEPP. As a final aim of the project, turbines will be designed for several potential hydro power plants and a data bank of hydro turbines will be formed based on head and flow rates. The data bank will include the design specifications, manufactured models and test results of the model hydro-turbines.

II. Design Methodology and Analyses

In this study, a CFD-based design method is used to obtain the runner blade shape and characteristics. Fig. 1 shows the runner design methodology previously developed by Kaewnai, S. and Wongwises, S. [6]. The process starts with the design of the runner blade with the supplied parameters for a specific power plant, Q (volumetric flow rate), H (Head) and N_s (Specific Speed). The shape and design for the runner blades alter with the changes in each of these parameters. According to these parameters, using in-house codes, basic runner angles of leading and trailing edges are determined. Runner blade shape is designed using a CFD software and the designed runner blades are meshed for Computational Fluid Dynamics (CFD) simulations using

Manuscript received November 30, 2012; revised February 3, 2013. This research is being supported by Turkish Ministry of Development.

F. Ayancik, U. Aradag, E. Ozkaya, K. Celebioglu, O. Unver, and S. Aradag are with the TOBB University of Economics and Technology, Ankara, 06560, Turkey (e-mail: fayancik@etu.edu.tr; uaradag@etu.edu.tr; eozkaya@etu.edu.tr; kutaycelebioglu@gmail.com; hounver@etu.edu.tr; saradag@etu.edu.tr).

DOI: 10.7763/IJMMM.2013.V1.35

the grid generation module of the same software. The geometric design obtained is simulated using CFD with k-ε turbulence model to obtain accurate results. If the design does not provide the necessary conditions, the procedure is repeated by changing the runner shape. When the designed shape provides the necessary conditions which are head, efficiency, outlet flow angle (alpha) and minimum pressure value for cavitation free operation; the CAD model of the blades is generated. Mechanical analysis of the design is also performed as a part of the developed runner design methodology. The best design is chosen after the mechanical analysis.

A. Solid Modeling of the Runner

Before the mesh generation for the design and CFD analyses, boundaries of the flow passage should be defined. BladeGen module of ANSYS was selected for the design because of its merits. BladeGen provides rapid simulations, modification and optimization when used with ANSYS CFX [7].

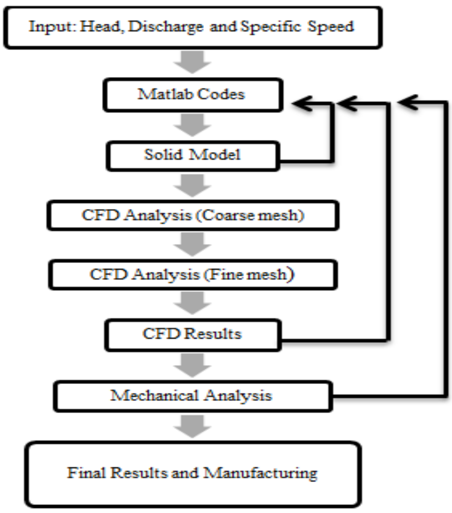

Fig. 1. Runner design methodology [6].

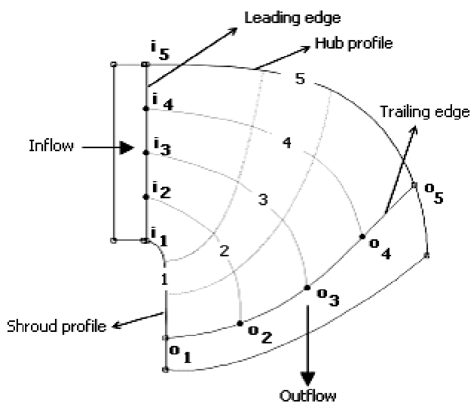

Fig. 2. Meridional sections of the runner blade [8].

BladeGen module defines the runner blade on a meridional plane. Fig. 2 shows the meridional plane and blade properties of the runner blade on meridional axis [8]. Blade parameters are defined for each of five meridional sections; these are blade thickness profile, blade angles and number of blades.

B. CFD Analyses of the Runner

Analyses of runner are performed based on the provided quantities for the specific power plant at hand: volumetric flow rate of 2.5 m^3/s, head of 43.75 m, circumferential speed of 1000 rpm, specific speed of 296 and 15 blades of runner. Number of blades, volumetric flow rate and circumferential speed parameters are used along with CFX turbo mode. Mass flow inlet and the pressure outlet are defined for the program.

C. Integrated CAD/CAM/CAE Environment for Collaborative Design and Manufacturing

Throughout the whole runner design methodology, we work in a CAD/CAM/CAE integrated environment in order to improve collaborative work and increase the blade quality while decreasing the time spent for the design and the manufacturing processes as also reported by [10]. The development stages are performed simultaneously by the collaboration of both design and manufacturing engineers. After the CFD analysis of the blade, the solid model is generated using CATIA V6, the Computer Aided Design program of Dassault Systemes and the mechanical analysis is performed by the same design environment [11]. As the solid model of the Runner Blade is generated, the same model is used to generate the NC-codes that are unique for the five axis milling machine that is going to be used for machining the meridional sections of the runner blade. The solid model is processed by DELMIA V6, the Computer Aided Manufacturing (CAM) tool of Dassault Systemes [12]. A five axis milling machine is crucial in order to machine the surface profile of the runner blade with highest precision. The five axis CNC milling machine in the Center for Hydro Energy Research is able to produce the full sized runner blades of medium capacity turbines and scaled down test blades of the models of high power turbines.

All of the tools that are used in the process, which are mentioned above as ANSYS, CATIA, DELMIA are installed on a single server computer. A Product Lifecycle Management (PLM) tool, named ENOVIA provided by Dassault Systemes as well, is integrated to the system, managing the lifecycle of model turbines from geometry generation to manufacturing and tests. ENOVIA will enable development of a model turbine engineering database which could accelerate geometry to-test cycle of new designs. The 3DEXPERIENCE Platform powered by ENOVIA will enable engineers and designers to leverage from the benefits of collaboration as it is robust enough to manage sensitive and mission critical data [13]. As an integrated solution of design, engineering and analysis, by focusing on design to product cycle, the delivery of critical information is available. Designers and engineers will be able to login to the system and design the runner blades collaboratively by dynamically using the data coming from the previous work and dynamically storing the data coming out of the new design process. Designers can also use another cluster computing environment as they perform CFD analyses that require high computational power. The CFD model prepared on the server computer is sent to ANSYS TurboGrid tool running on the high power cluster of TOBB ETU Center for Hydro Energy Research.

The CFD aided analysis of the runner blade has been completed, and solid models are present on the CAD/CAM environment. The last step of the runner design methodology will be completed by manufacturing the runner blade in the Advanced Manufacturing Laboratory of the Center for Hydro Energy Research.

III. Results and Discussion

Angles and initial dimensions of the blade profile were determined by the help of in-house MATLAB codes to create the runner blade profile. To obtain smooth profiles, lean and wrap angles definition were changed by checking the 3D blade profile and angles by the help of the ANSYS BladeGen module. Two different meshes are used for the CFD simulations [9]. Fig. 3 shows inlet and the outlet velocity triangles. Subscript 1 corresponds to inlet and subscript 2 corresponds to outlet.

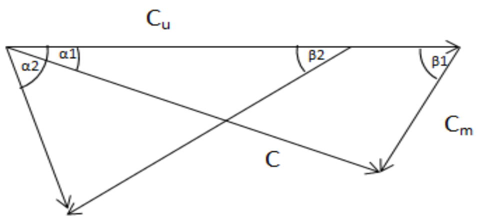

Fig. 3. Velocity triangles

Fig. 4 shows the velocity vectors and Fig. 5 shows the pressure distribution on the blade profile for the meridional plane. As shown in Fig. 4, swirl did not occur at the outlet of the runner blades as seen from velocity and pressure variations. Fig. 6 shows blade loading on runner. In Fig. 6, pressure decreases from the leading edge (LE) to the trailing edge (TE) as expected and as shown in Fig. 5. The LE and the TE are shown in Fig. 2. The value of zero on horizontal axis in Fig. 6 corresponds to LE and the value of one is TE.

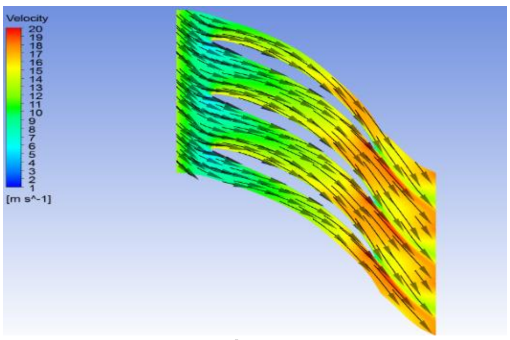

Fig. 4. Velocity vectors.

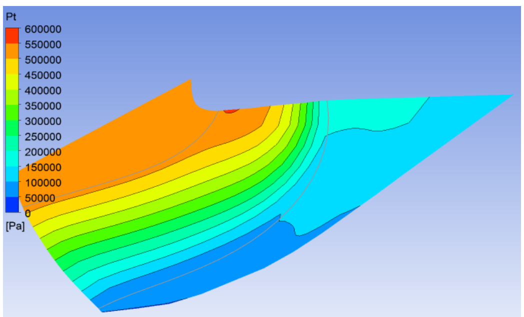

Fig. 5. Pressure distribution on meridional plane.

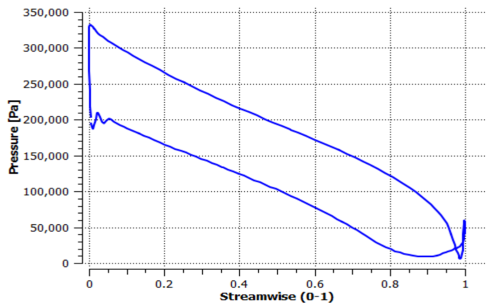

Fig. 6. Blade loading on runner

The pressure contour legend of Fig. 7 expresses that cavitation was reduced totally. The efficiency of the turbine designed in this study is 97.1%. When the losses of pressure and the other turbine components are taken into consideration, the efficiency is 92%.

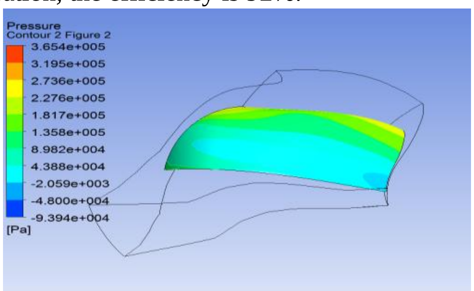

Fig. 7. Pressure distribution on runner blade.

Table I shows the properties of the fine mesh and Table II and Table III show the design results. In Table III, alpha expresses the angle of attack. It is the angle between the circumferential velocity and the incoming flow velocity. The angle alpha has the value of 28.6 degrees at outlet. In Table III, beta expresses the angle formed between the circumferential velocity and the meridional velocity in the rotational frame of reference.

TABLE I: Results From Blade Profile Mesh

Domains	Nodes	Elements
R1 Blade	508008	483840

TABLE II: Performance Results Table

Quantities	Values	Units
Rotation Speed	-104.72	[radian s^-1]
Reference Diameter	0.3918	[m]
Volume Flow Rate	2.5084	[m^3s^-1]
Head (IN-OUT)	43.7527	[m]
Shaft Power	1023670	[W]
Total Efficiency (IN-OUT) %	97.0709	

TABLE III: Summary Data Table

Quantities	Inlet	Outlet	Units
Total Pressure, P_t	528907	101126	[Pa]
Meridional Velocity, C_m	7.5509	6.2331	[m s^-1]
Circumferential Velocity, C_u	-17.3918	-5.8988	[m s^-1]
Flow Velocity, C	18.9603	10.2288	[m s^-1]
Flow Angle, Alpha	66.7354	28.5985	[Degree]
Flow Angle, Beta	-58.1923	-70.9669	[Degree]

Manufacturing ready technical drawings and solid model of the turbine is prepared and the next step is the manufacturing of this runner blade at TOBB ETU Center for Hydro Energy Research Manufacturing Laboratory as the first runner of the data bank of hydro turbines that will be developed in the near future.

IV. Conclusion

A collaborative design methodology is developed for the parametric, CFD aided design and manufacturing of hydro turbine runners. The aim of this study is to explain the design and manufacturing methodology for Francis type hydro turbines using a specific power plant in Turkey as a case study. The design of runner blade of Francis turbine to get the desired head and efficiency relied on the correction of runner shape with trial-error, in-house MATLAB codes and help of CFD. The efficiency for the designed runner at the Best Efficiency Point (BEP) is 92%. This designed runner will be the first runner manufactured and tested at TOBB ETU Center for Hydro Energy Research. The manufacturing and test facility of the center is still under construction and as a continuation of the work presented herein, the facility will start its operation in a short time.

Acknowledgment

The authors would like to thank to TEMSAN (Turkish Electromechanical Industry) and Su-Ener Engineering Inc. for their help and support. The computations are performed at TOBB ETU Center for Hydro-energy Research CFD Laboratory.

References

[1] R. R. Navthar, J. Tejas, D. Saurabh, D. Nitish, and A. Anand, "CFD analysis of Francis turbine," *International Journal of Engineering Science and Technology (IJEST),* vol. 7, no. 4, July 2012, pp. 3195-3197.

[2] G. I. Krivchenko, *Hydraulic Machines:Turbines and Pumps*, Moscow: Mir Publishers, 1986, ch. 2, pp. 22-28.

[3] K. Patel, J. Desai, V. Chauan, and S. Charnia, "Development of Francis turbine using Computational Fluid Dynamics," in *11st Asian International Conference on Fluid Machinery and 3rd Fluid Power Technology Exhibition*," India, 2011, pp. 1-3.

[4] P. Dirtina and M. Sallaberger, "Hydraulic turbines- basic principles and state-of-the-art computational fluid dynamics applications," in *Proc. Instn. Mech. Engrs,* vol. 213 Part C. 1999, pp. 85-102.

[5] K. Daneshkah and M. Zangeneh, " Parametric design of a Francis turbine runner by means of a three-dimensional inverse design method," in *25th IAHR Symposium on Hydraulic Machinery and Systems*, Timisoara, Romania, 2010, pp. 1-5.

[6] S. Kaewnai and S. Wongwises, "Improvement of the runner design of Francis turbine using Computational Fluid Dynamics," *American J. of Engineering and Applied Sciences,* vol. 4, no. 4, 2011, pp. 541-546.

[7] C. Buşea and S. Jianu, "Optimization of Axial Hydraulic Turbines Runner Blades Using Hydrodynamic Simulation Techniques," in *Proc. 6th International Conference on Hydraulic Machinery and Hydrodynamics,* Timisoara, Romania, 2004, pp. 69-71.

[8] G. Okyay, "Utilization of cfd tools in the design process of a Francis turbine," M.S.thesis, Dept. Civil Eng., Middle East Technical Univ., Ankara, Turkey, 2010.

[9] Y. Xiao, Z. Wang, Z. Yan, and J. Zhang, "Experimental and numerical analysis of pressure pulses characteristics in a Francis turbine with partial load," in *Proc. 25th IAHR Symposium on Hydraulic Machinery and Systems*, Timisoara, Romania, 2010, pp. 2-5.

[10] B.T. Lin and C.C. Kuo, "Application of an integrated CAD/CAE/CAM system for stamping dies for automobiles," *The International Journal of Advanced Manufacturing Technology*, vol. 35, issue 9-10, pp. 1000-1013, 2008.

[11] *CATIA V6R2013 User's Guide*, Dassault Systemes, 2013.

[12] *DELMIA V6R2013 User's Guide*, Dassault Systemes, 2013.

[13] *ENOVIA V6R2013 User's Guide*, Dassault Systemes, 2013.

Fatma Ayancık was born in Ankara. She graduated from Ankara Aydınlıkevler High School in 2006. She graduated from TOBB ETU Mechanical Engineering Department in 2012. Same year, she started her M.Sc. program at the same department on fluid dynamics with computational fluid dynamics (CFD) emphasis and she still continues her Masters study at same university. She is a member of Turkish Society of Mechanical Engineers.