TRANSACTIONS OF THE INSTITUTE OF FLUID-FLOW MACHINERY

No. 131, 2016, 111–120

Maciej Kaniecki[a], Zbigniew Krzemianowski[b*]

CFD analysis of high speed Francis hydraulic turbines

[a] *Research and Development Department, ZRE Gdańsk S.A., Litewska 14A, 80-719 Gdańsk, Poland*

[b] *The Szewalski Institute of Fluid-Flow Machinery of the Polish Academy of Sciences, Department of Hydropower, Centre for Hydrodynamics Fiszera 14, 80-231 Gdańsk, Poland*

Abstract

The paper presents the computational investigations of model Francis turbines (for high kinematic specific speed 95–115. Numerical calculations were carried out for flow systems including two different Francis runners. The computational fluid dynamics (CFD) analyses were performed for accurately mapped geometry of the laboratory test rig with installed model turbine. Commercial software was used for analysis.

Keywords: Francis turbine analysis; Inverse problem

Nomenclature

n_{sQ}	–	kinematic specific speed of hydraulic turbines and pumps
H	–	net head, m
Q	–	flow rate, m^3/s
n	–	rotational speed, rpm
I	–	turbulence intensity, %
$Y+$	–	dimensionless wall distance of the first grid cell
Y_k	–	guide vanes opening, [deg]
μ	–	water dynamic viscosity, kg/(m s)

*Corresponding Author. Email address: krzemian@imp.gda.pl

1 Introduction

Nowadays the hydroelectric power plants market has developed in two directions. From one side the brand new greenfield hydro projects are still the object of interest for large number of investors especially in the South America and Africa, and also modernizations of old fashioned hydropower plants units started to be the point of interests for lots of hydro companies. The second trend is very well noticed especially in Europe, where the potential for new projects is really limited. The hydromodernizations in Europe are very often directed to increase the overall parameters of turbines and extend the range of operating. An important aspect of modernization is also a cost reduction. Due to these requirements, the new designed turbines for refurbished hydropower plant must have a high level of efficiency in a wide range of work [1,2]. Additionally, the specific speed of the turbines, defined as

$$n_{sQ} = \frac{n\ Q^{0.5}}{H^{0.75}} , \tag{1}$$

where n is the rotational speed, Q is the discharge, and H is the net head, should be raised up due to the reduction of the turbines diameter. All of these requirements make the new projects very complicated, demanding a comprehensive approach to design issues.

In recent years ZRE Gdańsk S.A.was involved in a few modernization projects of hydropower plants. One of the market targets of hydromodernizations in Poland is replacement of low head single or twin old Francis turbines with highly efficient modern ones. For this reason, the company has been working with an experienced research partner (Institute of Fluid-Flow Machinery of the Polish Academy of Sciences – IFFM PASci) in order to develop a type series of high-speed Francis turbines.

This paper provides a summary of the work related to the numerical analysis of the flow system with Francis runner. Two different runners were designed, one prepared by ZRE Gdańsk and second prepared by IFFM. The idea was to design two runners with different specific speed fitted in the same constructions of the spiral case and draft tube. Wherefore, authors decided to use the same runner contours but different numbers and shapes of the runner blades. From the perspective of turbine manufacturer, such an action is meaningful, because the exchange of runner while maintaining the other elements will significantly expand the scope of the turbine applicability. The ZRE runner was characterised by specific speed of value 98 whereas the IFFM design was characterised by much higher specific speed runner of value 114. Both designs were subject to computational fluid dynamics (CFD) analysis. The calculations were performed for accurately mapped geometry of the laboratory test rig with installed model turbine. For the grid generation and CFD analysis commercial software was used. The analysis was conducted for turbine with different circumferential speeds and guide vanes openings. The results of CFD analysis have become the basis to select particular runner shape for further laboratory tests.

2 Input parameters of new design Francis turbines

As it was mentioned above the project of high-speed Francis turbine derives directly from the business needs of ZRE Gdańsk, performing modernization of hydropower plants. In a number of rehabilitation projects, realized by ZRE, it is justified to replace the old exploited Francis constructions with modern high-speed ones. Basing on the ZRE Gdańsk and IFFM experience the ranges of basic turbine parameters have been chosen. Due to capabilities of IFFM test rig it was decided to realize investigate the Francis runner model of 250 mm in diameter.

Table 1 shows the parameters characterizing the water turbine in the optimal operating point. The IFFM runner contains 11 blades and the rotational speed is 1275 rpm whereas the ZRE is equipped with 13 blades and the rotational speed is 1207 rpm.

Table 1: Input parameters for design operating point.

Parameter	Value
Rot. speed n, rpm	1200–1280
Net head H, m	12
Number of blades N	11–13
Thickness profile t	NACA 0010

The shape of runners and the other elements of the flow passage (spiral case, guide vanes and draft tube) were designed at the initial stage of the investigation and the design process will not be the subject of the paper. The input data were the geometry of designed passages alternatively with first and second runner.

3 Grid generation

The authors' intention was that CFD analysis should be fully homologous with the results of laboratory tests, hence the virtual model of the turbine flow system of the also contained the components of the test rig (intake pipeline and lower water container). The entire domain of the turbine passage was divided into six parts (inlet pipe, spiral case and stay vanes, guide vanes, runner, draft tube, and lower reservoir). The grid was generated separately in each part and combined interfaces [3].

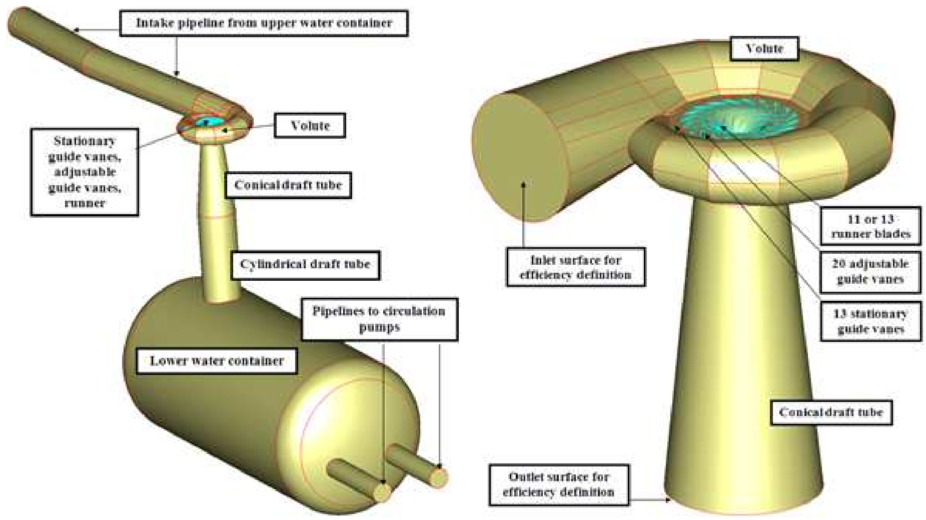

Figure 1: The views of the domain taken to the CFD computation (left) and domain taken to efficiency definition (right).

The grids of adjustable guide vanes (with 20 blades) and both runners were generated with the use of commercial software [8] NUMECA/AutoGrid5 . The mesh of the other domain elements was generated with the use of NUMECA/Hexpress. All the elements were of hexahedral shape. The Tab. 2 contains the basic information of the passage elements grid. The exemplary grids in the some regions of calculated domain is presented in Fig. 2. The total number of elements for the flow system with ZRE runner was 26.1 M, whereas with IFFM runner was 24.4 M. All interdomains were connected by the general grid interfaces (GGI), the runner-guide vanes and runner-draft tube connections were paid with special attention. The grid resolution in the draft tube was improved in the region of interface connections with the runner. The Y^+ values (Tab. 2) were determined for maximum guide vanes opening and optimal rotational speeds, n: for ZRE runner $n = 1207$ rpm and for IFFM runner $n = 1275$ rpm. The 90% of the mesh elements are characterized with orthogonality of over 0.6.

Table 2: Parameters of the turbine elements grid.

Element	Total number of elements	Y^+
Intake pipeline	0.274 M	1
Spiral case and stay vanes	7.850 M	1–3
Guide vanes – 20 blades	6.680 M	1–2
Runners – 11 (IFFM) and 13 blades (ZRE)	9.550/7.865 M	1–5
Draft tube	1.250 M	1–3
Lower water container	0.5 M	1

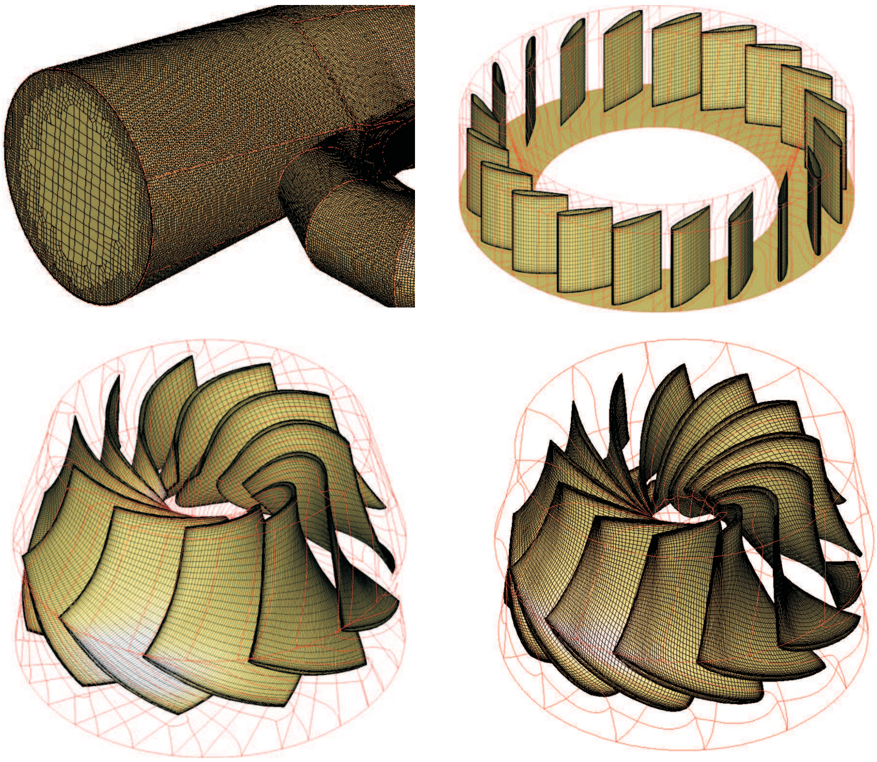

Figure 2: Grid distribution in the essential elements of the model turbine (inlet segment of spiral case, guide vanes, IFFM (left) and ZRE runners (right)).

4 Boundary conditions

At the inlet to flow domain for both analysed flow systems the total pressure was defined as 116872 Pa. This value includes gross head 12 m and is estimated on the basis of design dynamic pressure difference at inlet and outlet of flow domain. At the outlet the static pressure was assumed to be equal to zero (0 Pa). All the pressure values are referenced to the operating pressure – 101325 Pa.

The k-ω shear stress transport (SST) turbulence model was chosen to carry out the flow domain calculations. Turbulence parameters at inlet and outlet required were as follows:

inlet

- turbulence intensity: $I = 3.02\%$,
- hydraulic diameter: $D = 0.4$ m,

outlet (for a single outlet pipeline)

- turbulence intensity: $I = 3.12\%$,
- hydraulic diameter: $D = 0.257$ m.

Turbulence intensity I was calculated with the use of empirical formula derived from ANSYS/Fluent tutorial [9]

$$I = 16 \left(\underbrace{\frac{\rho\ V\ D}{\mu}}_{\text{Re}} \right)^{-0.125} = 16\ \text{Re}^{-0.125} , \tag{2}$$

where: V – inflow or outflow velocity (assumed on the basis of blade design), D – hydraulic diameter, ρ – water density (999.1 kg/m^3), μ – water dynamic viscosity (0.00116 kg/m s).

For the flow system with ZRE runner five guide vanes openings were assumed to calculations: $Y_k = 24°$, $27°$, $30°$, $33°$, and $36°$. For each of these openings six circumferential velocities of runner were performed: $n = 1147$, 1177, 1207, 1237, 1267, and 1297 rpm. For the flow system with IFFM runner six guide vanes openings were assumed to calculations: $Y_k = 24°$, $27°$, $30°$, $33°$, $36°$, $39°$. For each of these openings ten rotational speeds of runner were performed: $n = 1155$, 1170, 1185, 1200, 1215, 1230, 1245, 1260, 1275, and 1290 rpm.

The reason of assuming those values for both considered cases was obtaining the maximum efficiencies of flow systems depending on rotational speed and guide vanes opening.

5 Computational results

The calculations were carried out with the use of commercial software ANSYS/ Fluent 15 [9] (second order discretization, steady calculations, pressure-based solver). The optimal point of performance for flow system with ZRE runner was found for rotational speed $n = 1207$ rpm, guide vane opening $Y_k = 31°$ and flow rate: $Q = 0.245$ m^3/s. Analogically, for the flow system with IFFM runner the optimal point was found for rotational speed $n = 1275$ rpm, guide vane opening $Y_k = 35°$ and flow rate $Q = 0.284$ m^3/s. The hydraulic efficiency of the ZRE flow system achieved 92.3% whereas the IFFM flow system achieved 90.0%. The IFFM system was characterised by much higher flow velocities caused by higher flow rates what is the main reason of obtaining of lower efficiency.

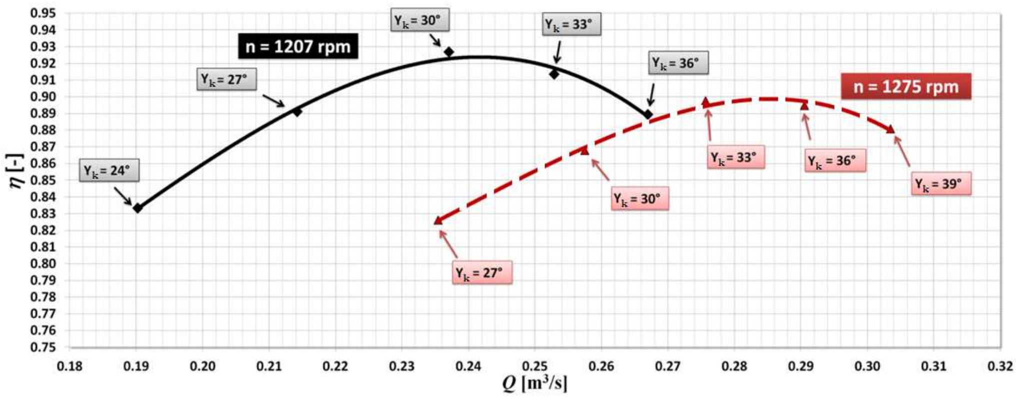

Figure 3: The CFD results of model Francis hydraulic turbine. Efficiency, η vs. flow rate, Q, for the runners designed by means of 2D inverse model (IFFM runner, dashed line) and 3D inverse model (ZRE runner, continuous line) with their optimal rotational speeds and Y_k – guide vanes opening.

In Figs. 4,5, and 6 some results of CFD calculations for the flow system for ZRE runner are presented. The results like pressure contours at suction side (related to operating pressure), velocity vectors according to tangential velocity at runner outlet, velocity magnitude contours at meridional plane of the draft tube are presented at optimal point of performance ($Y_k = 30°$) and off-design point (underloading) of operating ($Y_k = 24°$). As it can be seen, the tangential velocity components at the runner outlet for BEF (best efficiency point) are much lower than for off-design conditions. This confirms that the high efficiency of energy transformation in the runner is associated with reduction of angular momentum at the runner outlet. The nonuniform outlet velocity profile at off-design point propagates to the draft tube and generates extra losses in this region (Fig. 6, right view).

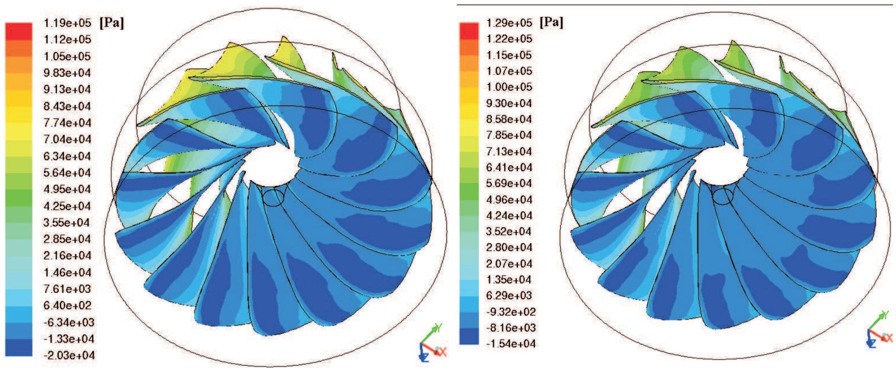

Figure 4: The CFD results of model Francis hydraulic turbine. Relative pressure contours at suction side of ZRE runner: : optimal point of performance (left view), off-design point of performance (right view).

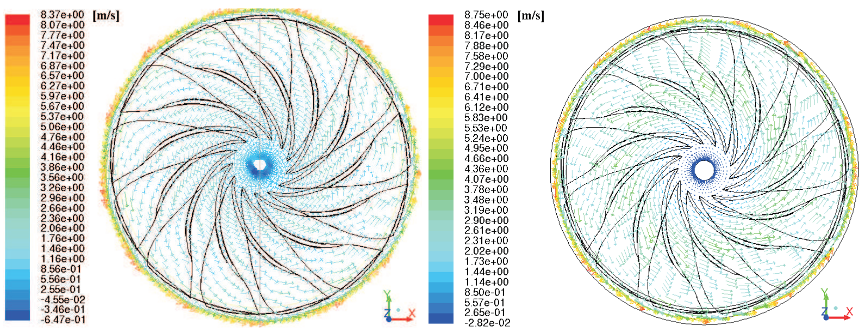

Figure 5: The CFD results of model Francis hydraulic turbine. Velocity vectors according to tangential velocity at outlet of ZRE runner. Left view: optimal point of performance; right view: off-design point of performance.

6 Summary and conclusions

The CFD calculations of the model test turbine were conducted for two flow passages, alternatively equipped with two runners designed by ZRE and IFFM,

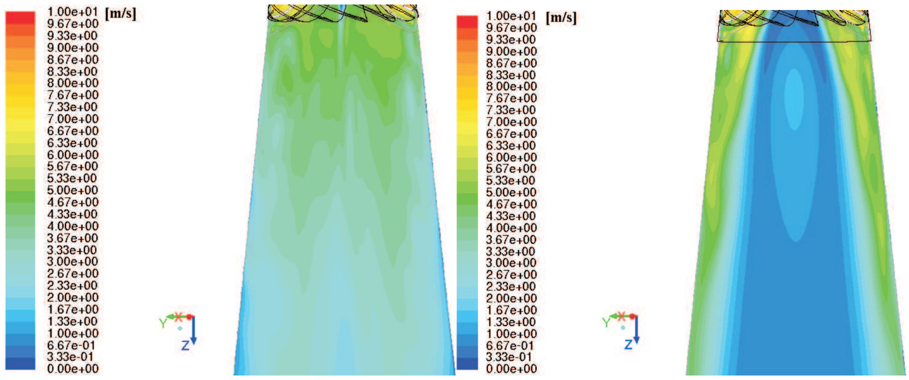

Figure 6: The CFD results of model Francis hydraulic turbine. Velocity magnitude contours at meridional plane of the draft tube (ZRE runner). Left view: optimal point of performance; right view: off-design point of performance.

respectively. Authors paid special attention for detailed copying of existed test rig geometry to model the conditions of future model tests. The calculations were realized for wide range of rotational speed to precisely investigate both runners operating parameters [4]. All calculated points were obtained based on the steady calculations (no transient stator-rotor interactions or vortex rope propagation was presented in the paper). Due to that the off-design operating points, especially in the underloaded regions were likely determined with the lower precision [5–7]. In the finalizing of this paper, the arrangements of transient calculations of hydraulic model turbine have been set. The numerical analyzes revealed that hydraulic efficiency of ZRE model turbine obtained 92.3% whereas 90.0% for IFFM design. Both design obtained relatively high value of design specific speed (respectively 98 and 114), which was the principal purpose of the project. Finally, based on CFD results, the ZRE runner has been chosen as the first for manufacturing and further laboratory tests. The IFFM runner will be tested afterwards. The final verification of the presented results for ZRE runner will be carried out during the model tests at the IFFM test stand.

Acknowledgements The paper was supported by the Polish National Centre for Research and Development and the Polish National Fund for Environmental Protection and Water Management under grant no. GEKON1/04/214228/27/2015 assigned for ZRE Gdansk and the Institute of Fluid-Flow Machinery in Gdansk (Polish Academy of Sciences). The numerical research was supported in part by PL-Grid Infrastructure.

Received in July 2016

References

[1] Drtina P., SallabergerM.: *Hydraulic turbines—basic principles and state-of-the art computational fluid dynamics applications*. Proc. IMechE **213**, Part C, 1999.

[2] Stick M., Michler W.: *Flexible turbine operation enabling frequency control*. Proc. HYDRO 2012, Bilbao 2012.

[3] Krane K.: *Simulations of the flow-driven rotation of the Francis-99 turbine runner*. M.Sc. thesis, Chalmers University of Technology Goteborg 2015.

[4] Stick M., Grunder R., Sallaberger M., Gehrer A., Parkinson E.: *How much complexity does your CFD simulation need?* [In :] Proc. 15 Int. Seminar on Hydropower Plants, Vienna 2008.

[5] Ruprecht A.: *Unsteady Flow Analysis in Hydraulic Turbomachinery*. [In :] Proc. 20th IAHR Symposium on Hydraulic Machinery and Cavitation, Charlotte 2000.

[6] Riedelbauch S., Klemm D., Hauff C.: *Importance of interaction between turbine components in flow field simulation*. [In :] Proc. 18th IAHR Symposium Hydraulic Machinery and Cavitation, Valencia 1996.

[7] Sick M., Casey M. V., Galpin P.: *Validation of a stage calculation in a Francis turbine*. [In :] Proc. XVIII IAHR Symposium on Hydraulic Machinery and Cavitation, Valencia 1996, 257-266.

[8] https://pl.scribd.com/document/89020372/Tutorial-Guide-AutoGrid-82-1-Advanced-Acrov5

[9] https://pl.scribd.com/doc/215349378/Fluent-14-5-Tutorial