CODEN STJSAO
ZX470/1327
ISSN 0562-1887
UDK 621.224:519.863:004.021:519.63:532.5(043)

Validation of Francis Water Turbine CFD Simulations

Zoran ČARIJA, Zoran MRŠA* and *Sanjin FUĆAK

Tehnički fakultet Sveučilišta u Rijeci
(University of Rijeka Faculty of Engineering),
Vukovarska 58
HR - 51000 Rijeka
Republic of Croatia

zcarija@riteh.hr

Keywords
CFD
Fluent
Francis turbine
Hydraulic characteristics
Hydraulic turbomachine

Ključne riječi
CFD
Fluent
Francisova turbina
Hidraulički turbostrojevi
Energetske značajke

Received (primljeno): 2007-05-05
Accepted (prihvaćeno): 2007-12-01

Original scientific paper

This paper compares data from calculated and measured results covering the whole operating range for a 20 MW Francis turbine in order to validate the CFD simulation. Computed hydraulic characteristics are determined for each analyzed operating point by running numerical simulations of turbulent fluid flow through a complete Francis Turbine model using the commercial fluid flow solver Fluent. The measured hydraulic characteristics were defined by on-site measurements according to the IEC 41 international field acceptance test standard. The computed characteristics show very good conformity of values and trend-lines with measured characteristics over the whole operating range. The deviations are lowest around the optimum and increase towards the boundaries of the operating range where the vorticity in the turbine diffuser is more significant. The aim was to analyze the behavior of the existing turbine and fine-tune the whole numerical experiment to achieve the level of accuracy necessary for a concept design of a revitalized turbine.

Validacija računalnih simulacija dinamike fluida na Francisovoj turbini

Izvornoznanstveni članak

U ovome radu prikazana je usporedba izračunatih i izmjerenih podataka 20 MW Francisove turbine u cijelom radnom području kako bi se validirala računalna simulacija. Izračunate energetske značajke su određene za svaku analiziranu radnu točku provođenjem numeričkih simulacija turbulentnog strujanja fluida kroz kompletnu Francisovu turbinu korištenjem komercijalnog fluid flow rješavača Fluent. Izmjerene energetske značajke određene su in-situ mjerenjima sukladno IEC 41 internacionalnom standardu. Izračunate značajke se vrlo slične, veličinom i trendom promjene, sa izmjerenima u cijelom radnom području. Najmanja odstupanja izračunatih podataka se nalaze u području optimuma, a povećavaju se prema krajevima radnog područja gdje postoji značajnije vrtloženje u difuzoru. Cilj je simulacije strujanja bilo analizirati ponašanje postojeće turbine i fino podesiti kompletan numerički eksperiment kako bi se postigla točnost neophodna za izradu idejnog rješenja revitalizirane turbine.

1. Introduction

Hydropower represents an almost ideal energy source with its flexible energy production and reservoir accumulation capabilities. The share in power production of so-called green energy can be increased by revitalizing existing power plants since there are important hydropower plants that were designed and constructed decades ago and many of them are not adjusted to the operating conditions that have been shaped throughout years of utilization.

It would be conceivable to increase the energy production output through renewable sources by 6-8 % through revitalization of such facilities. The revitalization of existing turbines represents the most cost-effective investment per unit of green energy produced, and further increases are possible through construction of small-scale and micro hydro turbine power plants.

Before computer simulations, hydraulic turbine shape was determined through experimental research using scaled-down turbine models. Such model research was expensive, allowing only minimal design variations while at the same time highly dependent on the skill and experience of researchers. The resulting shape did have better hydrodynamic properties but was still not necessarily the optimal shape.

The further development of numerical methods and an exponential increase of computing power in the last two decades have allowed the application of Reynolds time averaged Navier-Stokes equations (RANS, abbreviated) for fluid flow analysis in water turbines [5], [7], [13],

Symbols / Oznake

e_{hyd}	- specific hydraulic energy, m^2/s^2 - specifična hidraulička energija	μ	- dynamic viscosity, Pas - dinamička viskoznost
g	- acceleration due to gravity, m/s^2 - gravitacija	ρ	- density, kg/m^3 - gustoća
H_{loss}	- hydraulic losses, m - hidraulički gubici	φ	- general characteristic, - - općenita značajka
H_{st}	- static head, m - bruto pad	ω	- angular velocity, s^{-1} - kutna brzina
$HKSM$	- wicket gate servomotor position, % - hod klipa servo-motora		**Indeksi/Subscripts**
k	- turbulent kinetic energy, m^2/s^2 - turbulentna kinetička energija	1	- high pressure reference section - referentni presjek visokog tlaka
k_p	- pipeline constant, s^2/m^5 - konstanta dovodnog cjevovoda	2	- low pressure reference section - referentni presjek niskog tlaka
p	- pressure, Pa - tlak	HW	- headwater - gornja voda
M	- shaft torque, Nm - moment na vratilu	TW	- tailwater - donja voda
P	- runner mechanical power, MW - mehanička snaga na rotoru	t	- turbulent - turbulentni
Q	- volumetric flow rate, m^3/s - volumni protok	abs	- absolute - apsolutni
u	- velocity, m/s - brzina	tot	- total - totalni
Z	- elevation above sea level, m - nadmorska visina	st	- static - statički
	Grčka slova/Greek letters	calc	- calculated - izračunato
ε	- turbulent dissipation rate, m^2/s^3 - turbulentna disipacija	meas	- measured - izmjereno
η	- turbine hydraulic efficiency, % - hidraulička korisnost turbine		

[18], [19], [20] and [26] making fast analyses of many turbine operating points possible. Turbulent viscosity models (most often k-ε) were applied to better ascertain the complex character of fluid flow through the hydraulic system [5], [13], [15], [16], [20] and [22].

RANS analyses do not allow differentiation of the complete turbulent spectrum, unlike direct numerical simulations (DNS, abbreviated), which in turn require huge computational resources making them unviable for engineering purposes. Large eddy simulations (LES, abbreviated) have significantly lower computational requirements than DNS because they differentiate just one part of the turbulent spectrum producing very good results in many serious applications [8] and [24]. Constant advances in computer technology and parallel computing allow the application of LES models on real-world industry problems, but the steep resource requirements have confined its usage mainly to the academic community. LES applications on fluid flow in the space between runner blades of a Francis turbine give very good results compared to measured data. This shows the strong influence of the solid-fluid body interaction on the development and distribution of the flow field in the boundary layer [24].

Very large eddy simulation (VLES, abbreviated) is similar to LES but it differentiates a larger part of the turbulent spectrum and its applications on hydraulic fluid flow show an impressive accuracy compared to RANS turbulent viscosity models [3] and [4].

The most important research areas in hydraulic fluid flow concern the effect of cavitation [1], Francis turbine operation stability, interactions between movable and non-movable turbine parts, the fluid flow free-surface level phenomenon [14], [21] and [25] and the application of LES in numerical simulations [6] and [24]. Apart from individual scientific research, there are numerous international projects that gather scientists from various institutions, the most significant of which are GAMM, HPNURS, FLIDNT, HYDRODYNA, T99 [3], [18] and [23].

The purpose of this research is to develop a procedure for virtual revitalization. The procedure is general, performed entirely using computers and will be applied to the HPP Rijeka turbines. The interest in this particular case lies in the flood regimes of Rječina, during which the excess water is spilled (due to insufficient reservoir accumulation capabilities) when it could instead be harnessed using more powerful turbines. The following text contains comparisons of numerically computed and data measured in-situ for the case of a single mode of HPP Rijeka T2 turbine operation which are necessary to validate the procedure.

All simulations of fluid flow through HPP Rijeka turbines were performed using the commercial general purpose CFD software Fluent [12] on 4, 16 or 80-core multiprocessor computers at the Technical faculty of Rijeka.

1.1. HPP Rijeka

The dam constructed on the Rječina river created a reservoir with a usable volume of 420 000 m^3, harnessing the water potential for hydropower production. The maximum operating water level of the artificial lake is 229.5 m above sea level. The 3117 m long horizontal tunnel has a diameter of 3.2 m while the 803 m long penstock has a 2.2 m diameter at the end which forks asymmetrically in two branches that supply the turbines. The two Francis turbines are situated in the underground engine room at a height of 1.6 m above sea level. After passing through the turbine system, the water goes through the tailrace into the Rječina riverbed which leads into the Adriatic sea after 1 km. The total derivation length is 4300 m.

1.2. In-situ measurements of hydraulic characteristics

The hydraulic measurements which determined the hydraulic characteristics of HPP Rijeka were performed by the Zagreb Brodarski institute according to the 1991 IEC 60041 [11] regulations which reference the required ISO standards. Figure 1 shows the basic measurement points: 1 being the referential high-pressure section (turbine inlet) and 2 the outlet while HW and TW are the headwater and tailwater levels.

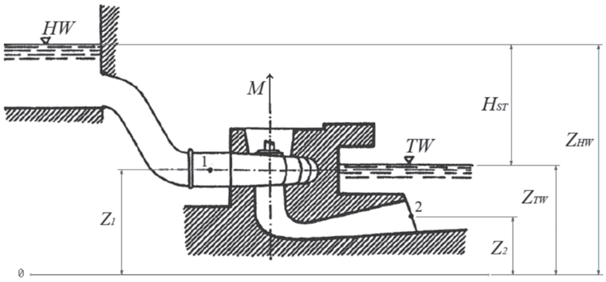

Figure 1. Measurement point schematic for Francis turbine hydraulic characteristics

Slika 1. Shematski prikaz mjerenja energetskih značajki Francisove vodne turbine

The measurements were performed in single (individual) and parallel (both) turbine operation modes at different headwater (Z_{HW}) levels in the reservoir and at different predetermined wicket gate positions. These positions were defined by the piston position of the wicket gate regulation servomotor (wicket gate fully closed at 0 %, and fully open at 100 %), or *HKSM* in the following text. Each operating point measurement lasted 20 minutes, of which the first 14 were spent changing and stabilizing the state in the hydraulic system of HPP Rijeka, and the last 6 minutes were used to gather data.

The significant values are calculated as follows:

$$e_{\mathrm{hid}} = \frac{p_{\mathrm{abs1}} - p_{\mathrm{abs2}}}{\rho} + \frac{u_1^2 - u_2^2}{2} + g \cdot (z_1 - z_2), \tag{1}$$

$$P = M \cdot \omega, \tag{2}$$

$$\eta = \frac{P}{\rho \cdot Q \cdot e_{\mathrm{hid}}}. \tag{3}$$

The following hydraulic characteristics were directly measured or derived from measured data: volumetric flow rate (Q), torque (M), turbine power (P) and turbine efficiency (η). They were subsequently compared to the values of corresponding characteristics computed from the results of performed fluid flow simulations.

The following variables were used for setting boundary conditions in the fluid flow simulations: *HKSM*, headwater level (Z_{HW}) and tailwater level (Z_{TV}).

The pipeline constant that was previously undetermined was extracted from measured data in order to evaluate the pipeline losses according to the expression:

$$H_{\mathrm{loss}} = k \cdot Q^2. \tag{4}$$

The value of tailwater level (Z_{TW}) is necessary to calculate static head but it is also necessary for setting proper outlet boundary conditions for the computational fluid flow simulations. The greatest contributor to tailwater level height, aside from the water level in Rječina riverbed, is the water flow through the hydraulic system. The relation between tailwater level and volumetric flow, shown in Figure 2, was used to fine tune the outlet boundary conditions in fluid flow simulations

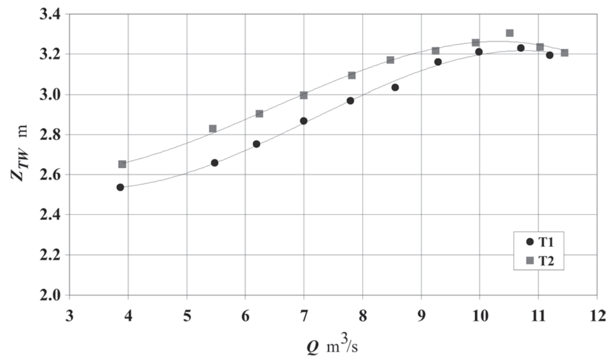

Figure 2. Tailwater level for individual operation of turbines T1 and T2

Slika 2. Visina donje vode pri pojedinačnom radu turbina T1 i T2

2. Geometry and numerical mesh

The case geometry represents the space between the two characteristic turbine sections: the inlet and the outlet. The inlet is the inflow section of the spiral casing and the outlet is the section joining the diffuser to the outflow channel. The ten stay-ring vanes of differing lengths and widths follow immediately after the spiral casing. Twenty equal guiding vanes are situated next in the radial direction of water flow, finely regulating its direction of impact on the runner blades. The runner consists of fifteen blades set between the hub and shroud. After the runner, the water flows through the vertical segment and the diffuser elbow, finally being transported through its horizontal section to the outflow channel.

The turbine geometry was CAD modeled using the commercial packages *Catia (Dassault Systemes) and Gambit (ANSYS Fluent Inc.)* mostly based on available documentation. The runner was an exception caused by the lack of appropriate technical documents and it was modeled according to a 3D scan of one blade of the spare replacement runner. Due to the significant complexity of the geometry with several rotating parts, it was necessary to model it partitioned separately into the spiral casing, the stay ring, the wicket gate, the runner and the diffuser. The partitions, as shown in Figure 3, were then assembled to create the complete turbine model, shown meshed in Figure 4. The position of the guiding vanes in the wicket gate was parameterized as a function of the wicket gate regulation servomotor position (*HKSM*) allowing a quick generation of the turbine geometry for any HKSM by simply changing that parameter.

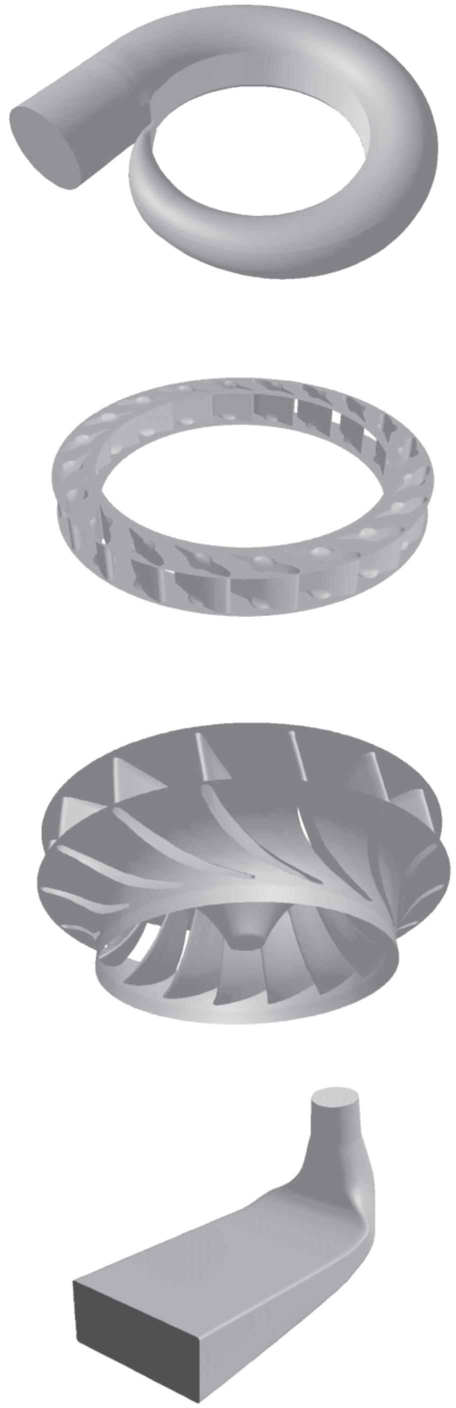

Figure 3. Partitions of the turbine geometry

Slika 3. Parcijalni dijelovi geometrije turbine

The numerical mesh was discretized in several adjoining parts according to the geometrical partitioning shown in Figure 3. Almost 95 % of the domain consists of hexahedral elements that ensure maximum stability and solution accuracy, while only the most complex parts of the geometry were meshed in tetrahedral elements, such as the spiral tail. Regardless of the type of elements used, a minimum of four layers of prismatic elements were created on all solid walls to ensure the applied turbulence model's criteria for proper boundary layer differentiation. The solution-dependent dimensionless parameter Y+ was used as a measure of numerical mesh quality during each simulation run and was kept in the recommended range of 30 to 60 throughout the domain. The complete turbine numerical mesh was assembled from partitions and non-conformal grid interfaces were defined in Fluent to circumvent the mesh differences on adjoining boundaries.

Figure 4 shows only the outer surface of the assembled discretized mesh for the complete turbine. In order to increase solution accuracy, the element density is notably higher in the mesh around the guiding vanes and the runner blades as they hold the greatest velocity and pressure gradients. The resulting mesh consists of 4.5 million cells.

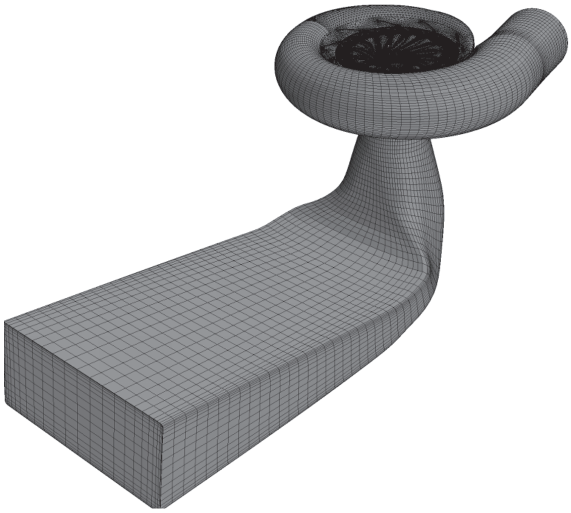

Figure 4. Numerical mesh surface of control volumes for the HPP Rijeka turbine

Slika 4. Mreža kontrolnih volumena turbine HE Rijeka (plošni prikaz); pogled sprijeda i straga

3. Computational models

3.1. Conservation equations

Once the Reynolds averaging approach for turbulence modeling is applied, the Navier-Stokes equations can be written for the mass and momentum conservation in Cartesian tensor form as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho \cdot u_i\right) = 0 , \tag{5}$$

$$\frac{\partial}{\partial t}\left(\rho u_i\right) + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho \overline{u_i' u_j'}\right). \tag{6}$$

The Reynolds stress term ($-\rho\overline{u_i' u_j'}$) is related to the mean velocity gradients by the Boussinesq hypothesis as:

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t \frac{\partial u_k}{\partial x_k}\right)\delta_{ij} . \tag{7}$$

The realizable k-ε turbulence model [10] with default model constants was used for turbulence closure with transport equations for the turbulent kinetic energy (*k*), and turbulent dissipation rate (ε). The turbulent viscosity μ_t is computed by combining *k* and ε as $\mu_t = \rho C_\mu k^2 / \varepsilon$ where C_μ is a function of the mean strain and rotation rates, the angular velocity of the system rotation and k and ε .

The realizable *k*-ε model has shown good performance applied on flows with strong streamline curvature, vortices and rotation, such as the complex fluid flow in the water turbine presented in this paper.

3.2. Solution algorithm

The above set of equations is solved in Fluent using standard finite volume techniques. Equations are integrated over the individual computational cells and, in the case of unsteady simulations, over a finite time increment. The 2nd upwind scheme was used to calculate convective flux on the boundary surfaces of control volumes. This 2nd order scheme is the least sensitive to mesh structure imperfections. Using 2nd order schemes is extremely important as it was observed that the reduced accuracy of 1st order schemes can lead to 5% lower computed turbine efficiency due to the erroneously computed pressure field.

The SIMPLE algorithm, which uses a relationship between velocity and pressure corrections, was used to enforce mass conservation and to obtain the pressure field.

3.3. Boundary conditions

Inlet

The *pressure inlet* boundary condition defining total pressure on the inlet section of the spiral distributor was selected as the domain inlet. Total pressure (p_{tot}) can be separated into static and dynamic components as follows:

$$p_{\text{tot}} = p_{\text{st}} + \rho \cdot \frac{u^2}{2} \,. \tag{8}$$

The values for those components are not defined before the actual simulation run, but the total pressure values can be rederived from the known headwater level according to the following expression:

$$p_{\text{tot,in}} = \rho \cdot g \cdot \left(\left(Z_{\text{HW}} - Z_1 \right) - H_{\text{loss,HW-in}} \right). \tag{9}$$

where $H_{\text{loss, HW-in}}$ represents fluid flow losses between the headwater level and the turbine inlet section.

As the volumetric flow rate is initially unknown, these equations are used to correct the total pressure values every 10 iterations using the currently computed volumetric flow rate values. The convergence criteria are met when the values for volumetric flow rate and total pressure stop changing with further iterations.

Outlet

The *pressure outlet* boundary condition is set on the outlet section with a defined static pressure corresponding to the hydrostatic pressure at the same location. The hydrostatic pressure is defined by tailwater level (Z_{TW}) expressed as a function of volumetric flow (Figure 2). The values are corrected during the iterative simulation run using the currently computed volumetric flow rate value.

All walls are defined as non-porous and with standard roughness, the fluid selected is standard water liquid and the gravity is included in the operating conditions.

3.4. Convergence criteria

The selected necessary convergence criteria were the maximum residual value of less than 10^{-5} (Figure 5) while the inlet volumetric flow rate and runner blade torque remain unchanged despite further iteration. Unchanging values for volumetric flow rate result in stable boundary conditions and allow the calculation of hydraulic characteristic for the selected operating point when the convergence criteria are met.

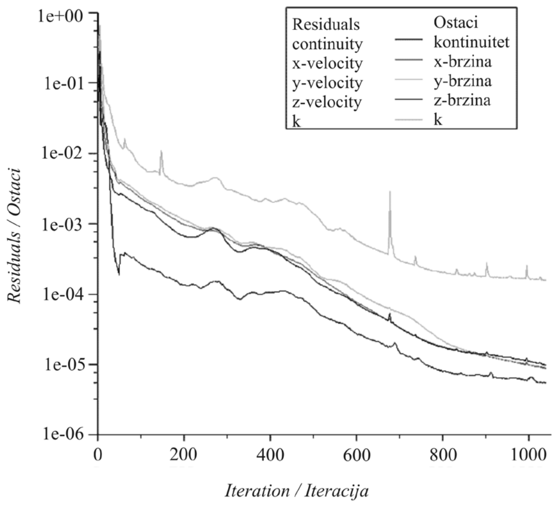

Figure 5. Residuals (a measure of discretized equation solution inaccuracy)

Slika 5. Ostatci (mjera netočnosti rješenja diskretiziranih jednadžbi)

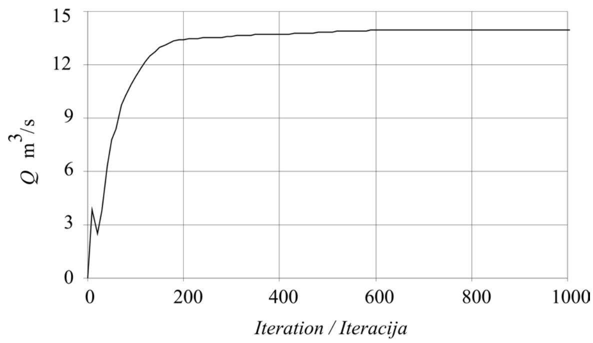

Figure 6. Convergence history for the inlet section volumetric flow rate

Slika 6. Povijest konvergencije volumnog protoka na ulaznom presjeku turbine

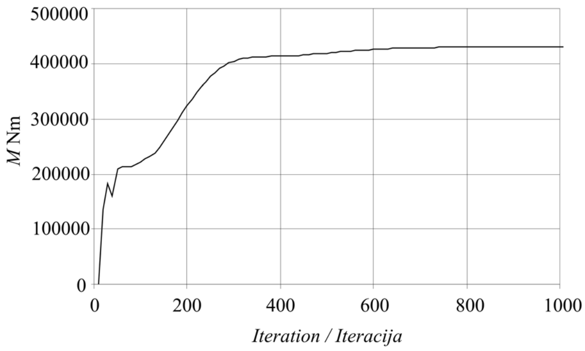

Figure 7. Convergence history for the turbine runner torque

Slika 7. Povijest konvergencije momenta na rotoru turbine

4. Results

Fluid flow simulations through HPP Rijeka turbines were performed for many operating points measured in-situ always taking into account the parameters of the hydraulic system which were determined prior to running the simulation. The relevant boundary conditions present during the measurements were reproduced in the simulations to ensure regularity of the comparison between computed and measured data. This included setting the proper headwater level (Z_{HW}; expression 9) and guiding vane position (defined by HKSM) for each analyzed operating point. The pipeline constant of the hydraulic system measures 0.033 [s^2/m^5]. The pipeline losses (expression 4,9) as well as the tailwater level were corrected during iteration based on currently computed volumetric flow rate (Figure 2). The hydraulic characteristics for the analyzed operating point were calculated upon meeting convergence criteria and a side-by-side comparison of their differences is shown in the following table and graphs.

The relative difference of computed and measured variable values was obtained through the following expression:

$$\Delta\phi_{\%} = \frac{\phi_{\mathrm{calc}} - \phi_{\mathrm{meas}}}{\phi_{\mathrm{meas}}} \cdot 100\ \%. \tag{10}$$

This difference is taken relatively to measured data difference as the measurements carry their own inaccuracy [11].

Table 1. Relative difference of measured and computed values; single mode T2

Tablica 1. Relativna razlika izmjerenih i izračunatih veličina; pojedinačni rad T2

HKSM [%]	$\Delta Q_{\%}$ [%]	$\Delta M_{\%}$ [%]	$\Delta P_{\%}$ [%]	$\Delta\eta_{\%}$ [%]
40	-1.18	-0.30	-0.25	1.35
50	0.30	1.50	1.56	1.62
60	-0.37	-0.21	-0.16	0.46
70	-1.51	-0.68	-0.62	1.11
80	-1.02	1.32	1.39	2.89

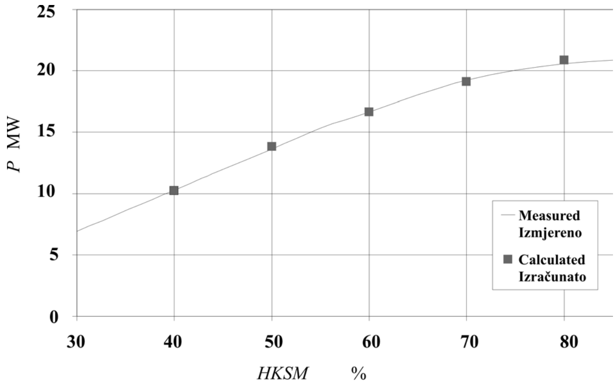

Figure 8. Comparison of computed and measured power over HKSM; single mode T2

Slika 8. Usporedba izračunatih i izmjerenih snaga u ovisnosti o HKSM; pojedinačni rad T2

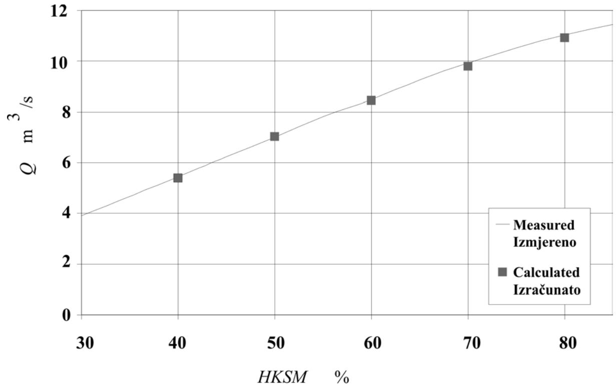

Figure 9. Comparison of computed and measured volumetric flow rate over HKSM; single mode T2

Slika 9. Usporedba izračunatih i izmjerenih protoka u ovisnosti o HKSM; pojedinačni rad T2

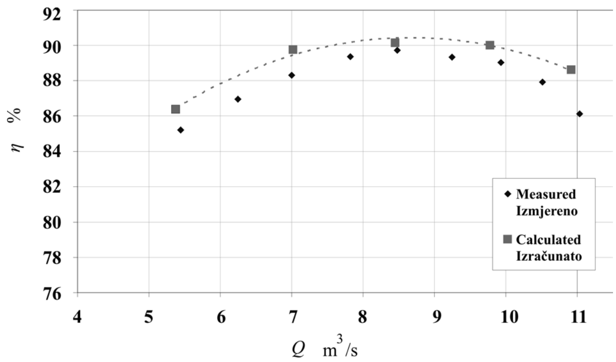

Figure 10. Comparison of computed and measured turbine efficiency over volumetric flow rate; single mode T2

Slika 10. Usporedba izračunatih i izmjerenih korisnosti turbine; pojedinačni rad T2

The largest deviations were expectedly present at the outer boundaries of the operating range as these are the positions furthest from the optimum and entail vortexes in the diffuser together with boundary layer separation zones. The task of accurately simulating fluid flow with larger zones of boundary layer separation at solid wall boundaries is exceedingly difficult regardless of the turbulent viscosity model and the size of the domain. It is therefore not surprising that the computed results at those operating points should differ more from the measured data than near the optimum efficiency point.

Despite the difference between computed and measured data, both computed and measured hydraulic characteristic values show very similar trends. The power and volumetric flow rate correspond very well, especially in the high-flow operating range where the slope of the curves decreases notably, as shown in Figures 8 and 9.

The computed turbine hydraulic efficiency is somewhat higher than the measured value and the difference increases towards the boundaries of the operating range, leaving the range around the optimum very similar as there are almost no vortexes in the diffuser. The turbine optimum efficiency point is almost at the same volumetric flow rate of 8.5 [m^3/s], with very similar curve trends in either direction away form the optimum, for both computed and measured data.

The inaccuracy of computed results for all analyzed characteristic values was within ±2 % over the whole analyzed operating range, increasing in accuracy towards the turbine optimal efficiency point to less than 0.5 % inaccuracy.

Besides computing hydraulic characteristics, CFD simulations also give a detailed insight into the complex structure of Francis turbine fluid flow, as shown in the following Figures 11 and 12.

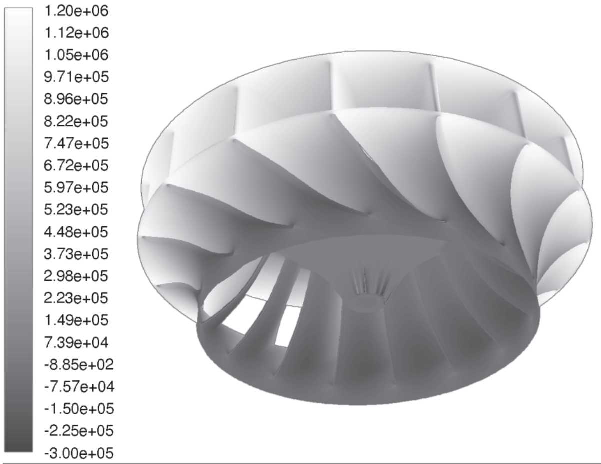

Figure 11. Contours of static pressure on runner surfaces

Slika 11. Konture statičkog tlaka na stijenkama rotora

The contours of static pressure visualize the pressure difference between the pressure side and the suction side of the runner blades. It is this pressure difference that creates the force resulting in torque. The static pressure contours clearly show the stagnation points (or stagnation curves in 3D) on which the complete energy of the fluid is converted into pressure energy.

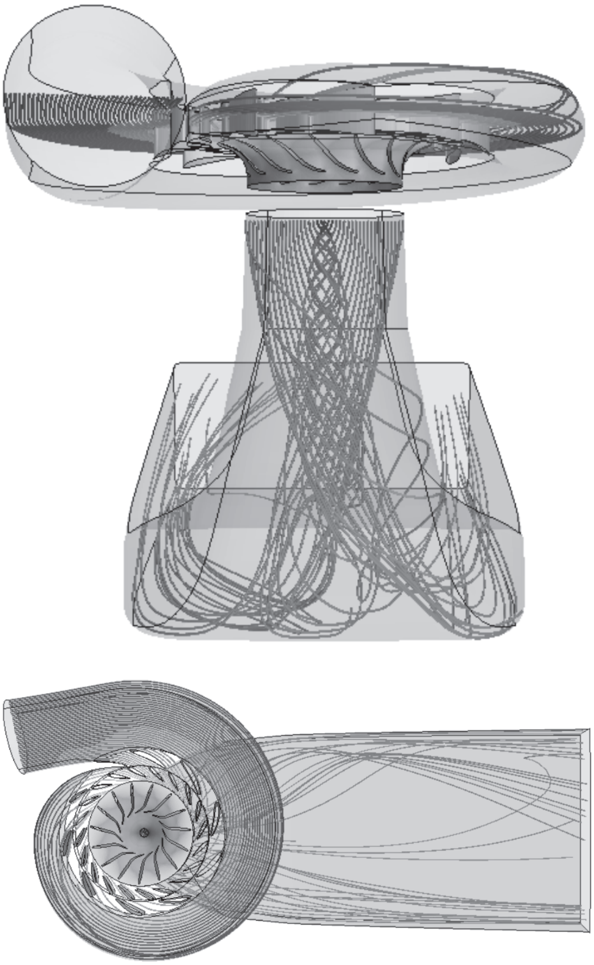

Figure 12. Fluid flow pathlines in the HPP Rijeka turbine, colored by total pressure, 100 % *HKSM*

Slika 12. Strujnice toka fluida u trubini HE Rijeka, obojane po totalnom tlaku, 100 % *HKSM*

The pathlines follow a very stable fluid flow in the spiral distributor and between the stay ring blades which turns into an exceptionally complex turbulent flow in the diffuser caused by out of optimum operation. This results in the kinetic energy of the fluid flow being partially transferred onto the diffuser. The elbow type diffuser additionally increases the fluid flow complexity by creating two dominant vortexes in its horizontal part (at 100 % *HKSM*).

5. Conclusion

The comparison of calculated hydraulic characteristics of HPP Rijeka T2 turbine in single mode shows minimal deviations between values obtained from computed and measured data. All computed hydraulic characteristics were obtained in CFD fluid flow simulations using the same boundary and operational conditions as those that were present during in-situ measurements.

The revitalization of outdated turbines is a procedure where minimal geometrical shape modifications of certain turbine parts lead to significant improvements in turbine characteristics. Such modifications of geometry are quick to perform on computer and the resulting modified turbine can be promptly analyzed using CFD simulations.

Assuming that the fluid flow simulations of the geometrically altered turbine have of the same degree of accuracy as the analyzed existing HE Rijeka turbine, it is possible to virtualize the whole process of revitalization by performing it entirely using computer simulations. This significantly reduces the duration and the cost of the revitalization process as it makes the expensive model manufacture and analyses unnecessary.

The turbine geometry was modeled according to available technical specifications enabling fluid flow simulations in single and parallel modes of operation across the whole operating range. The flow-dependent specifics of the hydraulic system in HPP Rijeka were ascertained from raw measurement data in the first phase of research. The computed hydraulic characteristics in the whole operating range conform very well with measured data. The largest deviations occur at the operating range boundaries furthest from the optimum efficiency point where vortexes occur in the diffuser as they are difficult to simulate accurately. Fluid flow simulations in operating points on the left leg of the efficiency curve are particularly problematic due to large separation zones occurring in the horizontal part of the diffuser. Computed result inaccuracy for all analyzed values was within ±2 % over the whole analyzed range with only minimal inaccuracy at the turbine optimum – less than 0.5 %. That successfully concluded the validation of the whole process presented in this paper for the purpose of developing a procedure for virtual revitalization which is complete and will be presented in a future paper.

REFERENCES

[1] AVELLAN, F.: *Introduction to Cavitation in Hydraulic Machinery*, Proceedings of the 6th International Conference on Hydraulic Machinery and Hydrodynamics, HMH2004, Timisoara, Romania, 2004.

[2] BREKKE, H.: *A Review of Some Dynamic Problems in Hydropower Plants*, IAHR WG1, 2003, Stuttgart.

[3] BUNTIC-OGOR, I.; GYLLENRAM, W.; OHLBERG, E.; NILSSON, H.; RUPRECHT, A.: *An Adaptive Turbulence Model for Swirling Flow*, Conference on Turbulence and Interactions TI2006, Porquerolles, France, 2006.

[4] BUNTIC, I.; HELMRICH, T.; RUPRECHT, A.: *Very large eddy simulation for swirling flows with application in hydraulic machinery*, Workshop on Vortex Dominated Flows - Achievements and Open Problems,Timisoara, Romania, 2005.

[5] ČARIJA, Z.; MRŠA, Z.: *Complete Francis Turbine Flow Simulation for the Whole Range of Discharges*, 4th ICCSM, Bizovac, Croatia, 2003.

[6] DAHLSTRÖM, S.: *Large Eddy Simulation of the Flow Around a High-Lift Airfoil*, PhD thesis, Chalmers University of Technology, Göteborg, 2003.

[7] DAVIDSON, L.: *An Introduction to Turbulence Models*, Chalmers University of Technology, 2001.

[8] DAVIDSON, L.: *LES-RANS of Channel Flow*, Chalmers University of Technology, Göteborg, Sweden, April 2000

[9] FERZIGER, J. H.; PERIC, M.: *Computational Methods For Fluid Dynamics*, ISBN: 3540420746, Springer-Verlag, 1996.

[10] Fluent 6.2 Documentation, *User's Manual*, Fluent Inc., 2005.

[11] IEC 60193, *Hydraulic turbines, storage pumps and pump-turbines - Model acceptance tests*, ISBN: 2-8318-4993-4, 1999.

[12] KRIVCHENKO, G.: *Hydraulic Machines: Turbines and Pumps*, ISBN 1-56670-001-9, CRC Press, 1994.

[13] LIPEJ, A.; POLONI, C.: *Design of Kaplan Runner Using Multiobjective genetic algorithm optimization*, Journal of Hydraulic Research, Vol. 38., 2000.

[14] MACK, R.; ASCHENBRENNER, T.; ROHNE, W.; FARHAT, M.: *Validation of bucket flow simulation using dynamic pressure measurements*, Proceedings of the 22nd IAHR Symposium on Hydraulic Machinery and Systems, Stockholm, Sweden, 2004.

[15] MAJI, P. K.: Biswas, *Analysis of Flow In The Spiral Casing Using a Streamline Upwind Petrov Galerkin Method*, International Journal for Numerical Methods In Engineering, 45, 1999.

[16] MARJAVAARA, B. D.; LUNDSTR T. S.: *Redesign of a sharp heel draft tube by a validated CFD-optimization*, International Journal for Numerical Methods In Fluids 2006, 50.

[17] MRŠA, Z.: *Optimal Design of Spiral Casing Tongue and Wicket Gate Angle by Decomposition Method*, International Journal for Numerical Methods, Vol. 17, 1993.

[18] NILSSON, H.; DAVIDSON, L.: *Validations and Investigations of the Computed Flow in the GAMM Francis Runner and the Hölleforsen Kaplan Runner*, Proceedings of 21st IAHR Symposium, 2002, Lausanne.

[19] NIČENO, B.; FRANKOVIĆ, B.: *Slobodna konvekcija u prostoru omeđenim horizontalnim kanalom i cijevi*, Strojarstvo, 39, 1997.

[20] NILSSON, H.; DAVIDSON, L.: *A Numerical Comparison of Four Operating Conditions in a Kaplan Water Turbine, Focusing on Tip Clearance Flow*, Proceedings of the 20th IAHR Symposium, 2000, U.S.A.

[21] PERRIG, A.; AVELLAN, F.; KUENY, J.; FARHAT, M.; PARKINSON, E.: *Flow in a Pelton Turbine Bucket: Numerical and Experimental Investigations*, Journal of Fluids, Volume 128, Issue 2, 2006.

[22] SOPTA, L.; MRSA, Z.: *Numerical modeling of three-dimensional flow in the spiral casing of a Francis turbine*, International Conference on Hydraulic Machinery, Ljubljana, 1988.

[23] VU, T.; NENNEMANN, B.; CIOCAN, G.; ILIESCU, M.; BRAUN, O.; AVELLAN, F.: *Experimental Study and Unsteady Simulation of the FLINDT Draft Tube Rotating Vortex Rope*, Proceedings of the Hydro 2004 Conference, Porto, Portugal, 2004.

[24] ZHANG, L.; GUO, Y.; WANG, W.: *Large eddy simulation of turbulent flow in a true 3D Francis hydroturbine passage with dynamical fluid–structure interaction*, International Journal for Numerical Methods in Fluids, 2007.

[25] ZHANG, Z.; PARKINSON, E.: *LDA application and the dual - measurement-method in experimental investigations of the free surface jet at a model nozzle of a Pelton turbine,* Proceedings of the 11th International Symposia on Applications of Laser Techniques to Fluid Mechanics, Lisboa, Portugal 2002.

[26] WOLF, I.; FRANKOVIĆ, B.; VILIČIĆ, I.; JURKOWSKI, R.; BAILLY, A.: *Analiza prijelaza topline na valovitom lamelnom orebrenju izmjenjivaca topline,* Strojarstvo, 46, 2004.