ANALELE UNIVERSITĂȚII
"EFTIMIE MURGU" REȘIȚA
ANUL XXIII, NR. 1, 2016, ISSN 1453 - 7397

Evaluation of Hydraulic Loads on the Runner Blades of a Kaplan Turbine using CFD Simulation and Model Test

Zoltan-Iosif Korka, Constantin-Viorel Campian, Daniel-Iosif Balint, Vasile Cojocaru, Ana-Maria Budai

CFD (Computational Fluid Dynamic) is today a standard procedure for analyzing and simulating the flow through several hydraulic machines. In this process, the fluid flow domain is divided into small volumes where the governing equations are converted into algebraic ones, which are numerically solved. Computational results strongly depend on the applied mathematical model and on the numerical methods used for converting the governing equations into the algebraic ones. The goal of the paper is to evaluate, by numerical simulation, the hydraulic loads (forces and torques on the runner blades) of an existent Kaplan turbine and to compare them with the experimental results obtained from model test.

Keywords: *CFD, Kaplan turbine, numerical simulation, QTurbo3D*

1. Introduction

The tendency of using computational fluid dynamics (CFD) in the area of fluid-rotor interactions, increased in the last years. It was brought in by Dietzen and Nordmann [1] in 1987, but because of the large computational time, has not been extensively used in the past. The first CFD utilizations to rotor dynamics were noticed in the area of hydrodynamic bearings and seals. Lately, CFD has become a research tool for studying the fluid-rotor interactions in centrifugal pumps [2]. Recently, CFD has become more common in research, design and development of hydraulic machines. In this respect, Ruprecht et al. [3], [4] used CFD to estimate forces and pressure pulsations within Francis and axial turbines.

Campian and Nedelcu [5] have investigated the flow between the runner blades and the runner chamber of a Kaplan turbine, obtained results being compared with those resulted on the model and prototype.

Balint and Campian [6] present a complete methodology of the hydrodynamic design for the runner of axial hydraulic turbines (Kaplan) using the finite element method. The procedure starts with the parametric design of the meridian channel. Next, the stream traces are being computed in the meridian channel using the finite element method. The finite element method was implemented numerically in original software called QTurbo3D.

Cojocaru et al. [7] have simulated the tip vortex position and the source of the cavitation erosion of a Kaplan turbine. The simulation results performed on the Kaplan turbine runner blade have shown a decrease of the pressure variation at the lip tips of the modified lips, compared to the original ones. This decrease was shown by the limit values of the pressure coefficient, as well as by the gradient of the variation.

Without claiming to have exhausted all the examples, Jošt et al. [8] have done a comparison between numerical simulations and measurements of a six-blade Kaplan turbine, in order to establish an appropriate numerical setup for accurate and reliable simulations of Kaplan turbines. Values of torque, losses and discharge, obtained by different turbulence models, were compared to each other and to the measurement results.

The benefits of using CFD to calculate the hydraulic loads of hydraulic turbines have not yet been fully explored. In the present work, CFD is used to evaluate the hydraulic loads (forces and torques) on the runner blades of an existent Kaplan turbine and to compare them to the experimental measurements performed on the model.

2. Case study

The goal of this research was to analyse the hydraulic loads (forces and torques on the runner blades) of an existing Kaplan turbine. For this purpose, it was chosen the operating point of the turbine having the characteristics shown in Table 1.

Table 1.

Characteristic	Symbol	Value
Net head	H_n	27,1 m
Turbine output	P	182,8 MW
Runner blade opening	φ_{PDF}	12,78°
Guide vane opening	α	38,75°

The passage of water flow through the turbine generates on the blades a resultant force R, which can be decomposed into a tri-orthogonal coordinate system as shown in Figure 1.

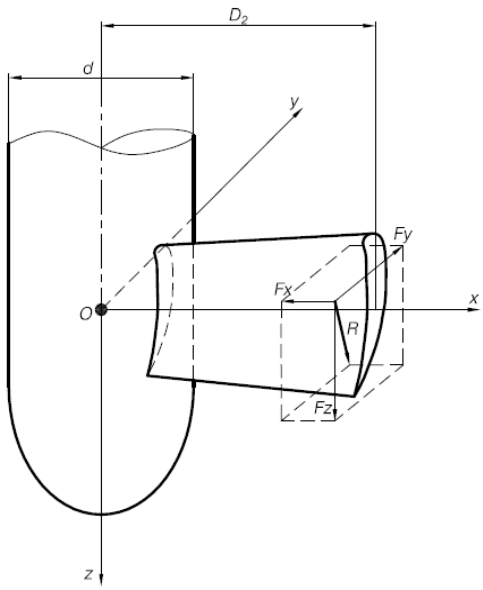


Figure 1. Forces acting on the runner blades

The resultant force R and the corresponding torque M can be expressed by following equations:

$$\vec{R} = \overrightarrow{F_x} + \overrightarrow{F_y} + \overrightarrow{F_z}\ , \tag{1}$$

$$\vec{M} = \overrightarrow{M_x} + \overrightarrow{M_y} + \overrightarrow{M_z}\ , \tag{2}$$

where:

F_x- the force along the turbine blade axis;
F_y- the force which gives the driving torque;
F_z- axial thrust force (acting on the axis of the runner);
M_x- hydraulically torque, which acts on the runner blades;
M_y- bending moment of the blade;
M_z- driving torque.

3. Modeling and simulation of the flow through the turbine

For the present work, the blades are analyzed by using the QTurbo3D software. As presented by Balint et al. [9], this in-house developed software can be used for design, analyze and optimization of the blades from axial hydraulic turbo machines (Kaplan + bulb, Francis and Darrieus).

The software has been successfully used in other researches performed within Centre for Research in Hydraulics, Automation and Thermal Processes (CCHAPT) from "Eftimie Murgu" University of Resita, as for example for the flow investigation on a Kaplan turbine with modified cross section [10], for the hydrodynamics optimization of the runner blades from reaction hydraulic turbines [11] or for the constructive improvement of the blades from a Kaplan turbine [12].

In the present research, QTurbo3D was implemented for reading-in the points of the runner blade surface. This interface is shown in Figure 2.

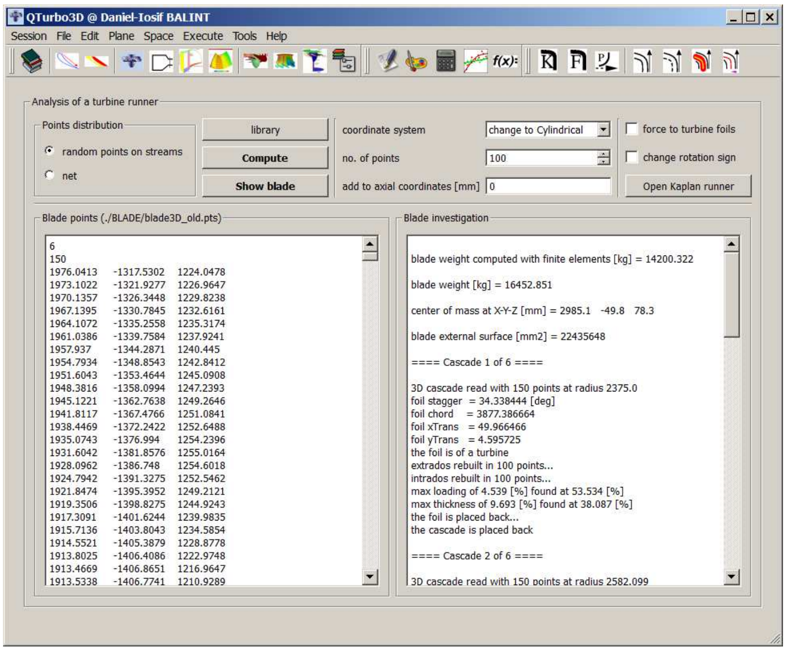
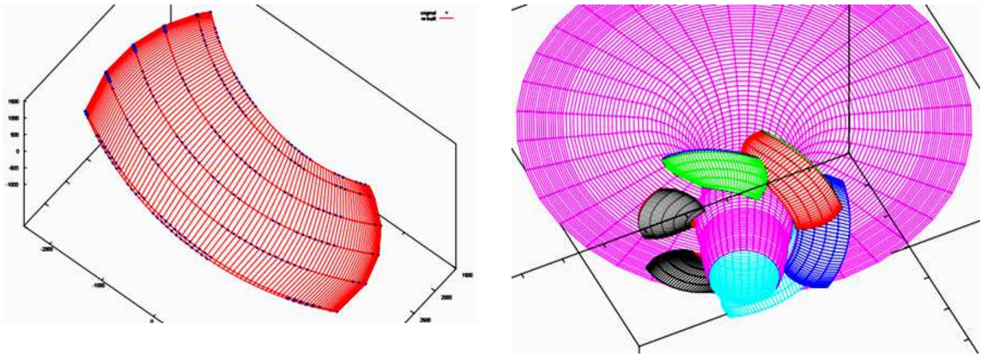


Figure 2. QTurbo3D interface for reading-in the points of the runner blade

The points were placed on circular stream tubes, the procedure being common to any topology of hydraulic runner blade (axial or radial-axial). For this purpose, six sections were used, each of them with 150 points. The points were extracted from the geometry built in SolidWorks using the information of the measured runner model.

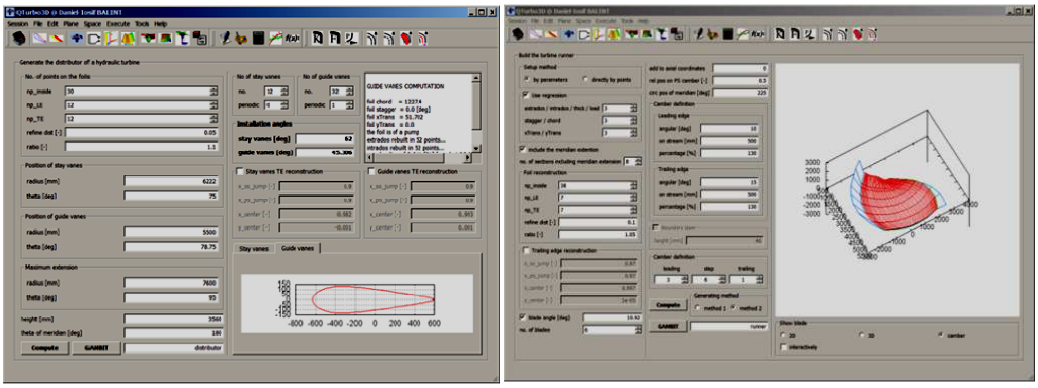
Each section of the runner was re-built in 2x100 points (refined at leading/trailing edge positions), for the pressure side and suction side, respectively as presented in Figure 3. The QTurbo3D software analyzed separately each section delivered and next it generates an output with the info of the cascade foils found: stagger angle, chord length, position of the axis relatively to the chord of the foil, value and positions of the maximum thickness and curvature of each foil.



a. b.

Figure 3. Reconstruction of runner sections
(a- pressure side; b- section side)

Further, QTurbo3D plots the re-built blade, using the library of Gnuplot distribution. Also, the presence of hub surface helps the user to better understand the shape and orientation of the runner blades in the assembly.



a. b.

Figure 4. Generating one periodic domains with QTurbo3D software
(a- guide vane; b- runner blade)

The computation of the flow is performed using Fluent/ANSYS distribution, while the geometry is built in Gambit pre-processor of Fluent package. QTurbo3D generates the scripting files for building automatically the computation domains as presented in Figure 4 a. for the guide vane and Figure 4 b for the runner domain.

The geometry of the distributor and the runner is built in Gambit/ANSYS using the interface of QTurbo3D. The resulted computational domain can adapt automatically to the wicket gate opening and configuration.

Furthermore, the computational domain of the runner adapts automatically to the runner opening and position of distributor-runner interface.

For the present research, one single periodical domain of the guide vane and one single periodical domain of the runner are used as presented in Figure 5 a.

The flow in these domains is solved coupled using the mixing interface algorithm implemented in Fluent/ANSYS distribution. The grid size for this one-to-one channel is around 1,1M hexahedral cells. All the scripting command files are generated by QTurbo3D software for both distributor and runner domains. At the inlet of cylindrical surface of guide vane, the flow velocity and turbulence is imposed. The flow is passed from the guide vane domain toward the runner domain through the mixing interface algorithm which averages circumferentially the flow velocity angles and turbulence quantities. The static pressure profile at guide vane outlet and the velocity profile at runner inlet are computed internally by this mixing interface method applied onto a conical geometric surface sloped at 15° in meridian view of the hydraulic channel of the turbine (Figure 5 b).

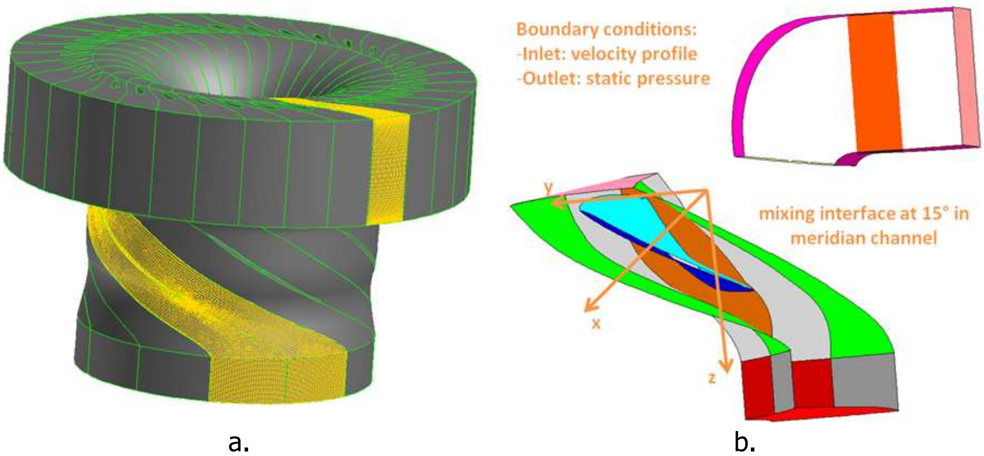


Figure 5. Conditions of numerical simulation
(a- Periodicity of the channels; b- Axis system in the domain orientation)

At the outlet of the runner domain, the pressure profile is used together with the radial pressure equilibrium condition implemented in Fluent/ANSYS software. This condition is suitable numerically for regions with reduced radial flows, which is also the case of runner outlet.

The flow is assumed turbulent and the log-law model is used to solve the flow at the fluid-solid interfaces.

The Renormalization Group turbulence model (RNG) implemented in Fluent/ANSYS was used, together with 2nd order pressure discretization and 3rd order momentum discretization for the internal equations. Steady state of the flows in the absolute reference frame of the distributor and relative reference frame of the runner was solved, together with a continuum hydrodynamic transfer through the mixing interface between these domains.

Furthermore, the intersection between the force ray and a particular Cartesian plane (oriented along any coordinate axis) is also delivered by Fluent/ANSYS. Therefore, in order to quantify the force application point onto the runner blade, only 2 points are required to trace the force ray in Cartesian system, as presented in Figure 6. Two points at ±2 [m] axial location are used to get the final position of the hydrodynamic force onto the runner blade.

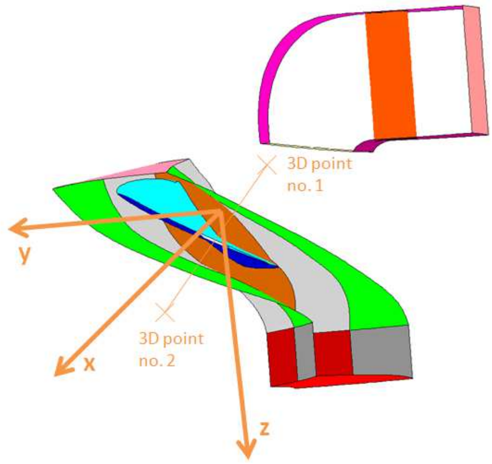


Figure 6. Tracing the force ray onto the runner blade

The hydraulic loads on the Kaplan runner blade obtained by CFD calculation are shown in Table 2.

Table 2

Hydraulic force on runner blade				Hydraulic torque on runner blade		
$F_{x,CFD}$ [kN]	$F_{y,CFD}$ [kN]	$F_{z,CFD}$ [kN]	R_{CFD} [kN]	$M_{x,CFD}$ [kNm]	$M_{y,CFD}$ [kNm]	$M_{z,CFD}$ [kNm]
-34,1	1.149	1.862	2.188,2	-542,6	-6.051	4.075

4. Experimental results and discussions

Experimental evaluation of the hydraulic loads was performed on the turbine model, installed on the test rig, using the same operating conditions. The obtained results are shown in Table 3.

Table 3

Hydraulic force on runner blade				Hydraulic torque on runner blade		
$F_{x,MT}$ [kN]	$F_{y,MT}$ [kN]	$F_{z,MT}$ [kN]	R_{MT} [kN]	$M_{x,MT}$ [kNm]	$M_{y,MT}$ [kNm]	$M_{z,MT}$ [kNm]
Not measured	Not measured	1.930,1	2.119,9	-520	Not measured	Not measured

Usually, on model tests (MT) there are measured the following hydraulic loads on the runner blades:

- axial thrust force F_z, which acts on the blades generating the power on the shaft, parameter which is used in the calculation of the axial thrust bearing of rotor shaft;
- resultant of hydraulic forces R;
- hydraulically torque M_x, acting on the runner blades, value which is used in the calculation of the guide vanes.

As shown in Figure 7, excepting the axial thrust force F_z, the CFD calculation gives higher hydraulic loads compared to the model test (MT).

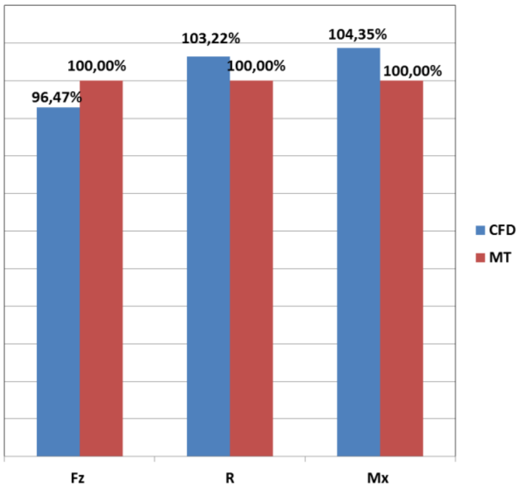


Figure 7. Hydraulic loads obtained by CFD versus model test (MT) (percentage differences)

It can be noted that following percentage differences between numerical simulation and experimental measurements were obtained:

- axial thrust force F_z, obtained by CFD simulation is 3,53% lower (68,1 kN in absolute value), compared to model test result;
- resultant of hydraulic forces R from CFD simulation is 3,22 % higher (68,3 kN in absolute value) , compared to model test result;
- hydraulically torque M_x is 4,35 % higher (22,6 kNm in absolute value) in CFD calculation , compared to model test result.

Furthermore, it can be observed that the differences between the results obtained by numerical simulation and the experimental measurements are less than 5%, which confirms the current trend in using CFD as a faster and less expensive tool in the development of the hydraulic turbines.

5. Conclusions

The flow in the runner of a Kaplan turbine was solved by numerical computation. For this study, QTurbo3D software was used to link the whole methodology with Gambit+Fluent/ANSYS distributions.

CFD calculation of hydraulic loads on Kaplan turbine provides all the components of the hydraulic loads. Using these results, information necessary in the design processes can be obtained.

The results of hydraulic loads obtained from CFD are very close to the results obtained from model tests. Therefore, it can be concluded that the performance of the turbine can be best evaluated by CFD calculation, as experimental evaluation involves higher time and financial resources.

Another advantage of CFD calculation is that the experimental results can be checked if there is a geometrical similarity between numerical and experimental geometry.

Acknowledgement

The work has been funded by the Sectoral Operational Programme Human Resources Development 2007-2013 of the Ministry of European Funds through the Financial Agreement POSDRU/159/1.5/S/132395.

References

[1] Dietzen F. J., Nordmann R., *Calculating rotor dynamic coefficients of seals by finite-difference techniques*. Journal of Tribology, vol. 109, no. 3:pp. 388–394, 1987;
[2] Suzuki T., et al., *A numerical analysis of rotor dynamic forces on an artificial heart pump impeller in whirling motion*, Proceedings of the 12th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, Honolulu, Hawaii, USA, February 2008.
[3] Ruprecht A., et al., *Unsteady forces on the blading of an axial turbine caused by stator-rotor interaction*, Proceedings of the 9th IAHR WG International Meeting on the Behavior of Hydraulic Machinery under Steady Oscillatory Conditions, Brno, Czech Republic, September 1999.
[4] Ruprecht A., et al., *Numerical simulation of a complete Francis turbine including unsteady rotor/stator interactions*, Proceedings of the 20th IAHR Symposium on Hydraulic Machinery and Systems, Charlotte, NC, USA, August 2000, pp. 1-8.
[5] Campian C.V., Nedelcu D., *Cavitation tip clearance. Numerical simulation and experimental results*, 3rd Workshop on Vortex Dominated Flows, Timisoara, Romania, 1-2 June 2007.

[6] Balint D., Campian C.V., *Hydrodynamic design of axial hydraulic turbines*, Proceedings of the 4th WSEAS International Conference on Finite Differences- Finite Elements- Finite Volumes- Boundary Elements, pp. 104-109, 2011.
[7] Cojocaru V., et al., *Numerical analysis of flow in Kaplan turbine runner blades anticavitation lip with modified hydrodynamic profile*, Annals of "Eftimie Murgu" University from Resita, vol. XVIII, no. 1, pp. 229-234, 2011.
[8] Jošt D., et al., *Improvement of Efficiency Prediction for a Kaplan Turbine with Advanced Turbulence Models*, Strojniški vestnik - Journal of Mechanical Engineering, vol. 60, no.2, pp. 124-134, 2014.
[9] Balint D., et al., *QTurbo3D in-house software for hydrodynamics optimization of the Kaplan turbines from Iron Gates I hydro power plant*, Annals of "Eftimie Murgu" University from Resita, vol. XX, no. 1, pp. 179- 183, 2013.
[10] Cojocaru V., et al., *Numerical investigations of flow on the Kaplan turbine runner blade anticavitation lip with modified cross section*, Recent researches in mechanics, pp. 215-218, 2011.
[11] Balint D., et al., *Hydrodynamics optimization of runner blades for reaction hydraulic turbines*, IOP Conference Series: Earth and Environmental Science 15, 2012.
[12] Miclosina C.O., et al., *A method to combine hydrodynamics and constructive design in the optimization of the runner blades of Kaplan turbines*, IOP Conference Series: Earth and Environmental Science 15, 2012.
[13] Mulu B.G., et al., *Simulation-based investigation of unsteady flow in near-hub region of a Kaplan Turbine with experimental comparison*, Engineering Applications Of Computational Fluid Mechanics, vol. 9, Issue: 1, pp. 139-156, 2015.

Addresses:

- Lect., PhD. Eng. Zoltan-Iosif Korka, "Eftimie Murgu" University of Reşiţa, Piaţa Traian Vuia, nr. 1-4, 320085, Reşiţa, z.korka@uem.ro
- Prof. Dr. Eng. Constantin-Viorel Câmpian, "Eftimie Murgu" University of Reşiţa, Piaţa Traian Vuia, nr. 1-4, 320085, Reşiţa, v.campian@uem.ro
- PdD. Eng. Daniel- Iosif Balint, "Eftimie Murgu" University of Reşiţa, Piaţa Traian Vuia, nr. 1-4, 320085, Reşiţa, d.balint@uem.ro
- Lecturer, PhD. Eng. Vasile Cojocaru, "Eftimie Murgu" University of Reşiţa, Piaţa Traian Vuia, nr. 1-4, 320085, Reşiţa, v.cojocaru@uem.ro
- Lect., PhD. Eng. Ana-Maria Budai, "Eftimie Murgu" University of Reşiţa, Piaţa Traian Vuia, nr. 1-4, 320085, Reşiţa, am.budai@uem.ro