

Hydraulic Design of an Axial Runner Using Classical Methods and CFD Simulation

Peter Hlbočan^{1,*}, Marta Murgašová¹, Zoltán Csuka¹, and Michal Varchola²

¹ZTS – Výskum a Vývoj, a.s., Ul. Lieskovec 598/75, 018 41, Dubnica nad Váhom, Slovenská republika

²SjF STU v Bratislave, Námestie slobody 17, 812 31, Bratislava 1, Slovenská republika

Abstract. The paper deals with the principles of hydraulic design of runners of axial turbines or pumps. The procedures of classical methods of hydraulic design of primary runner geometry and the possibilities of subsequent optimization of this geometry by CFD simulation are presented. In the paper, the advantages of the procedure of hydraulic design of the primary runner geometry of a tubular Kaplan turbine with the application of the classical methods and the subsequent CFD simulation are shown on a specific example. The purpose of the CFD simulation is to verify the achievement of the required parameters of the designed runner geometry and its possible fine-tuning and optimization.

Keywords: *primary geometry; axial runner; hydraulic turbine; hydraulic design; CFD simulation*

1 Introduction

Hydraulic design of the axial turbine (or pump) runner (axial machine) is a challenging task that can be approached in several ways. The requirement of the hydraulic design is to achieve the required values of flow rate (discharge) and head at the given rotating speed at the best efficiency point (BEP). In addition, e.g.: runner diameter, cavitation properties, noise and vibration level, etc. may be required. Several hydraulic design methods have been developed based on simplified models of two-dimensional flow of an ideal liquid using the theory of flow around aerodynamic (hydrodynamic) profiles, for which experimentally determined courses of lift and drag coefficients are known. The measured lift and drag values refer to an isolated airfoil (they are obtained by measurements in wind tunnels), therefore it is necessary to apply correction coefficients processed in the form of diagrams, through which the lift coefficient of an isolated airfoil is corrected to a profile in an axial flow blade cascade. The mentioned methods strongly depend on the results of previous experimental research and optimization tasks. Based on them, the initial (preliminary) design of the main dimensions of the runner (diameter of the hub and chamber, blade span angle, etc.) and the efficiency of the machine is obtained. Calculation of the geometry of blade cuts (profiles) is carried out in the next step using the methods described above. In this way, the geometry of the runner is obtained, the parameters of which, as a rule, approximately correspond to the design requirements. We call this geometry primary geometry.

¹ Corresponding author: peter.hlboacan@ztsvav.eu

Subsequently, it is necessary to optimize the primary geometry in order to get closer to the design requirements or in order to achieve maximum efficiency values, etc. In the past, the optimization of the primary geometry was mainly carried out using experimental research, whereas the very important role played the experience of the researchers performing the task or knowledge regarding general relationships between hydraulic properties (performance and effective parameters) of the machine and geometry.

Currently, the optimization of the geometry is usually carried out on the basis of CFD simulation. A larger number of optimization algorithms are known, which are commonly applied to solve these tasks. Many of them are today a common part of commercial CFD (CAE) software (gradient methods, response surface DOE optimization, six sigma analysis, genetic algorithms, etc.). Of course, the essential question remains how to design the preliminary (i.e. initial – primary) geometry of the designed machine. In principle, it is possible to choose an "ad-hoc" primary geometry and, on the basis of an optimization algorithm, gradually and systematically modify the geometry to "find" the desired shape. The number of design points – geometric alternatives (and thus the optimization time) can be extremely high in this case. The second option is to start from the known geometry of the machine with similar parameters as specified in the hydraulic design requirements. The third option is to carry out the hydraulic design of the primary geometry using the classical method, which significantly reduces the number of design points (geometric alternatives) within the subsequent optimization algorithm. With the application of the mentioned optimization procedures, it is possible to almost "automatically" design the optimal geometry of the axial runner. A common feature of all three listed approaches to "automatic" optimization of the runner geometry is a relatively high number of geometry design points, which usually ranges from tens to hundreds.

So-called genetic algorithms are a specific group of current optimization algorithms. Multi-objective genetic algorithms (MOGA) are included, for example, in commercial software CAESES. This environment can be directly connected to the ANSYS Workbench software environment.

An example of the optimization of a Kaplan turbine with a three-blade runner using MOGA algorithms in the CAESES and ANSYS environment is described in the paper [14]. The primary geometry was designed and fully parameterized in the CAESES environment. Sensitivity analysis in the first phase of optimization revealed a greater impact of changing some basic parameters of the geometry. The SOBOL algorithm included in the CAESES environment was chosen for the sensitivity analysis. Totally 16 parameters defining the runner blades and 1 parameter describing the position of the distributor vanes were analyzed. As part of the sensitivity analysis, a total of 300 simulations took place, on the basis of which the dependencies of changes in individual parameters and their mutual correlations were determined. The most important parameters were the angle of the distributor vanes, the inlet and outlet angle of the runner blade and the runner blade chord length [14].

The MOGA algorithm was implemented for the optimization. Totally 9 parameters were optimized. Two objective functions were defined, for which the global minimum value was sought in the optimization process. They were $F_1=1-\eta$ (hydraulic efficiency) and $F_2=\sigma_{ref}$ (cavitation number at reference flow). The goal was to find the optimal ratio of the values of both objective functions. The initial hydraulic efficiency (of the primary geometry) was 90 %. The cavitation properties reached a value of $\sigma_{ref} = 1.25$. A similar hydraulic efficiency was achieved with a significant improvement of cavitation properties to $\sigma_{ref} = 0.83$ already by the sensitivity analysis procedure. The optimization process further increased the hydraulic efficiency to $\eta = 90.7$ % while achieving the cavitation coefficient $\sigma_{ref} = 0.77$. Approximately 700 simulations were carried out as part of the optimization process. The last phase of the optimization process was the verification of the

final geometry in the wider operating range of the heads and rotating speeds of the machine [14].

Paper [15] has a similar character. MOGA algorithms were used in the work [15] in the CADDRUN-opt environment (originally designed to optimize Francis turbines). Some problems are mentioned, because of which the automatic optimization of the blade geometry of the Kaplan turbine can be more difficult in the work [15]. One of them is dual turbine regulation, which increases the number of simulations needed. The second problem is the presence of a thin gaps between the runner blade tips and the chamber. The flow in these gaps has a significant effect on the hydraulic efficiency of the machine. That is why these gaps in CFD simulations are important, which is supposed to increase the complexity of creating a computational mesh and at the same time increases the need for CPU time or computing performance.

The first step of the optimization is the parameterization of the geometry (Fig. 1). In a given optimization phase, the geometry of the blades is changed, while the geometry of the runner hub and the turbine chamber remains unchanged. On the one hand, the parameterization of the shape of the runner blade should allow fully flexible variation of the complex hydraulic design. On the other hand, the parameterization should contain the smallest possible set of parameters to reduce the number of design points. Totally 24 geometric parameters were optimized: 16 parameters of blade geometry, 8 parameters determining the overall dimensions of the blade [15].

The objective functions were the requirements for hydraulic efficiency and cavitation properties, the value of which was maximized or minimized. The hydraulic efficiency value was monitored both in BEP ($Q_{11}^{(1)}, n_{11}^{(1)}$) and in full load mode ($Q_{11}^{(2)}, n_{11}^{(2)}$). The cavitation properties of the runner were estimated by calculating the torque increment in the region where the pressure p is lower than the saturated vapor pressure p_v . The goal was to minimize the relative area of cavitation W_{cav} on the suction side of the blade. The paper presented the achieved values of the hydraulic efficiency and the cavitation parameter described above after optimization for two different Kaplan turbines. The first of them had 5 blades and was designed for a head of 20 m. The optimization was solved using a multi-objective genetic algorithm (MOGA). Totally 25 populations were calculated, each consisting of 120 unique design points. In the next step, several suitable geometric designs (design points) are selected from the Pareto front for further analysis [15].

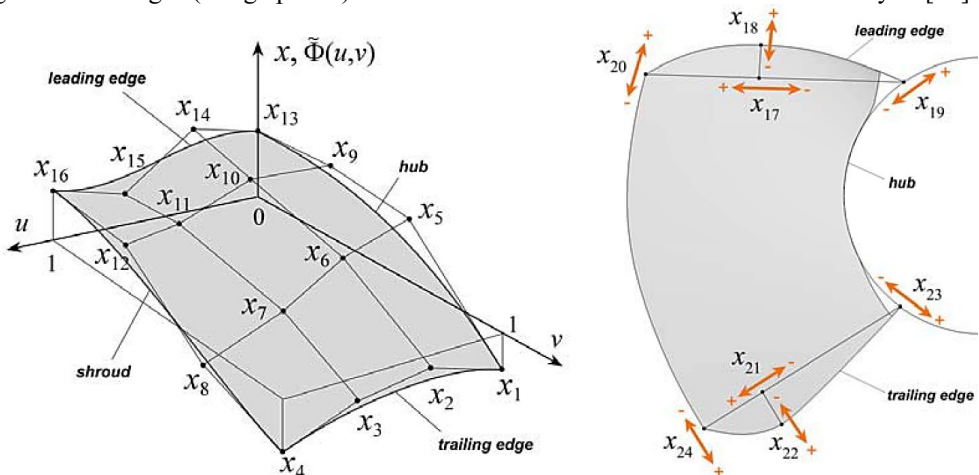


Fig. 1. Parameterized geometry of the runner blades [15].

Based on the presented results, it can be concluded that the automatic optimization was able to increase the efficiency to approximately 0.78% in BEP and to 0.76% in the full load

operating point. The cavitation properties of the optimized blade were also improved compared to the original blade [15].

The paper also presents the results of the automatic optimization of the Kaplan runner for a turbine with a head of 40 m and a runner blade number of 6. The efficiency results were again presented for the BEP and for the full-load operating point of the turbine. There were 29 populations during the optimization. From the calculated Pareto front, a specific geometry was chosen for further analysis that met the set of requirements. Based on the presented results, it can be concluded that the efficiency in BEP increased by approximately 0.15 % and the efficiency for the full load operating point increased by approximately 0.9 %. The cavitation properties have also improved significantly [15].

An alternative to automatic optimization algorithms is the so-called "engineering" approach which also utilizes CFD simulation as the tool for predicting parameters or hydraulic properties of the designed machine. Previous knowledge about the relationship between hydraulic properties (power and efficiency properties) of the machine and its geometry is utilized to a significant extent. The design of the initial - primary geometry is basically the same as for the automatic optimization approaches mentioned above. Geometry modifications in specific design points are carried out "manually". In this, the experience and knowledge of the designer play an important role. Analyzes of the internal flow pattern (distribution of velocity and pressure fields) can be an additional design tool. This is implemented as part of the post-processing of the individual geometry alternatives (design points) of the proposed machine, and based on them, it is possible to identify possible critical places (eddies, flow separation areas, etc.). Energy dissipation usually occurs in these places which results to reducing the efficiency of the proposed machine. Next geometry alternatives (design points) might be adjusted to eliminate or at least reduce the extent of identified problematic areas. The advantage of this approach is a significantly smaller number of design points compared to the "automatic" optimization algorithm.

The aim of the contribution is to show (using a specific example) the advantage of the design procedure of a tubular Kaplan turbine runner primary geometry using the classical method with the subsequent CFD simulation aimed at verifying the fulfillment of the required parameters and possible "fine-tuning" of the machine geometry. The main advantage of applying procedure in the design of the runner geometry is a principal reduction of the number of design (shape) alternatives of the proposed machine.

2 Classical methods of hydraulic design of runners of axial turbines or pumps

Let's consider a specific example from practice. The request was to design a turbine for a specific location according to the assignment (Fig. 2). The red frame in Fig. 2 indicates the required operating range of the turbine, and the purple line indicates the head value at which it is necessary to guarantee the performance parameters (power and efficiency). The required rotating speed is $n = 360 \text{ min}^{-1}$ and the diameter of the runner is $D = 1100 \text{ mm}$. Co. ZTS VaV, a.s. has its own portfolio of model turbines. When applying the "most suitable" model turbine from the current portfolio, it is possible to "cover" the expected operating range as shown in Fig. 2 on the right. The problem is low efficiency or failure to meet the required performance parameters with a guaranteed head and small flow rates (efficiency at the minimum flow rate should be at least 87%). From this reason, there was a request to modify the geometry of the original turbine, or to design a new turbine, which would have shifted the operating range towards smaller heads (flow rates) and increased efficiency in the lower left part of the operating chart. One of the ways to achieve this is to ensure a shift of the best efficiency point (and thus the entire operating chart) in the above-mentioned direction.

The best efficiency point of the original turbine is at a head of approx. $H = 10.75$ m. On a modified turbine, the best efficiency point should be approximately at a head of approx. $H = 8.2$ m, while the flow rate at the best efficiency point may remain unchanged, or may be a few percent smaller. It is important to increase the efficiency at the guaranteed head so that the efficiency at the minimum operating flow rate is at least 87 %.

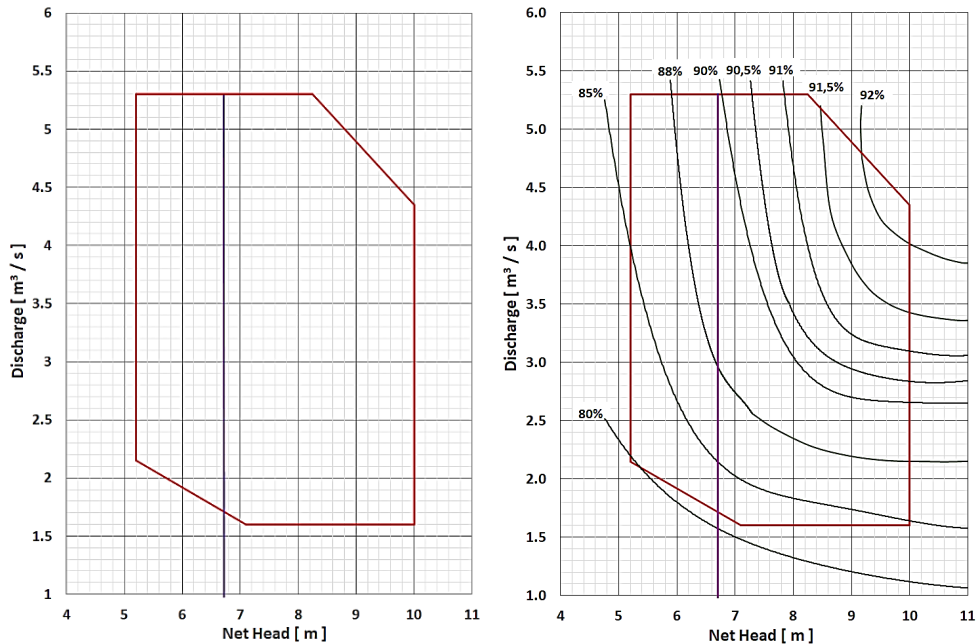


Fig. 2. Required operating range (left), operating chart of the original turbine with marked operating range (right).

In this case, the modification of the geometry of the original turbine was performed by the hydraulic design of the new runner using the "classical" approach with the application of the Voznesensky method. The geometry of all stator parts of the turbine remained original. Subsequently, CFD analysis was applied for the purpose of more detailed verification of the achievement of the required performance parameters.

It was based on the following required parameters at the best efficiency point: $Q = 4.36$ m^3/s , $H = 8.2$ m, $n = 360$ min^{-1} , $D = 1100$ mm.

The uniqueness of the presented procedure is that the runner of the Kaplan turbine was designed as the impeller of an axial pump. The fact that every hydrodynamic machine is in principle reversible and can work in a pump or in a turbine operating mode was used. The performance parameters at BEP of the machine are different for the pump and for the turbine operation mode. The ratio between the nominal flow rate or the nominal specific energy of the machine in pump and turbine operation mode varies in principle from machine to machine. Nevertheless, it is possible to observe certain tendencies that this ratio acquires depending on the achieved hydraulic efficiencies of the machine as well as on the specific speed [1, 2, 5].

If we can successfully estimate the specific values of the ratio between the nominal parameters of the machine in pump and turbine operation mode, we can perform a recalculation of the design (required) parameters of the machine in turbine mode to pump mode and apply these parameters as input parameters for the calculation of the axial runner geometry [1, 2]. This procedure was also applied in this case.

2.1 Hydraulic design of the axial runner meridional cut (flow path)

The procedure of the hydraulic design of the meridional cut is explained in more detail in the works [1 – 13]. The scheme of the meridional cut of the axial runner with the main dimensions is shown in Fig. 3 (left). The calculation is performed according to calculation relations in the Table 1.

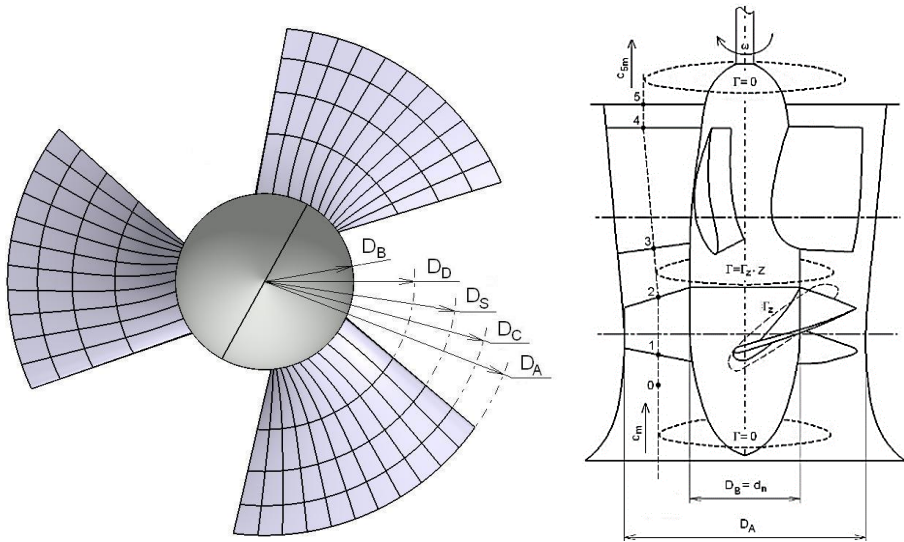


Fig. 3. Meridional cut, runner blades and streamline-cylindrical surfaces, on which the design of blade profiles is performed [3].

Table 1. Equations applied in the design of the meridional cut.

Coefficient of specific speed	$n_b = n \cdot \frac{\sqrt{Q}}{(g \cdot H)^{0.75}}$	(1)
Coefficient	$K_m = 0.0688 + 0.733 \cdot n_b^{1.1}$ <i>valid for: $0.33 < n_b < 1$</i>	(2)
Meridional velocity	$c_m = K_m \cdot \sqrt{2 \cdot g \cdot H}$	(3)
Runner chamber diameter	$D = D_A = \sqrt{\frac{4}{\pi} \cdot \frac{1.04 \cdot Q}{c_m} \cdot \frac{1}{1 - \left(\frac{D_B}{D_A}\right)^2}}$	(4)
Ration between the runner hub and the chamber	$\frac{d_n}{D} = \frac{D_B}{D_A} \doteq 0.63 - 0.346 \cdot (n_b - 0.25)$	(5)

2.2 Hydraulic design of axial pump runner blade profiles

The hydraulic design of blade profiles is usually realized on several cylindrical surfaces evenly distributed in the space of the runner (Fig. 3 on the right). The calculation itself utilizes the experimental results obtained in the wind tunnel on isolated profiles. The details of the calculation are explained in more detail in the works [1 – 6].

Blade number (z) is chosen mainly according to the specific speed of the machine (parameter n_b). Regarding the design of the number of blades, it is necessary to calculate a suitable blade span angle φ_r (Fig. 4 and Fig. 5).

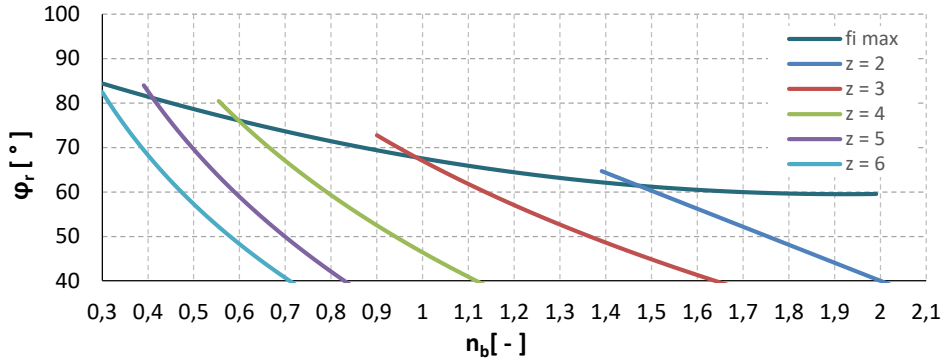


Fig. 4. Nomogram for determining the number of blades and the blade span angle with the marking of the upper limit curve.

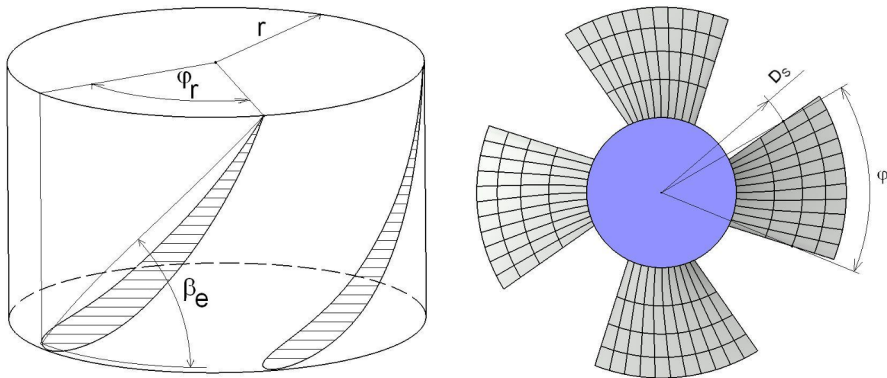


Fig. 5. View of the cylindrical surface of the blade cut [3].

Fig. 6 shows the velocity triangles, kinematic parameters and basic geometric parameters of the axial blade cascade of the runner, essential from the point of view of the achieved machine parameters. The task is to design these parameters as well as some other geometrical parameters of the blade profile so that the required performance or hydraulic parameters of the machine are achieved. The design itself is done iteratively.

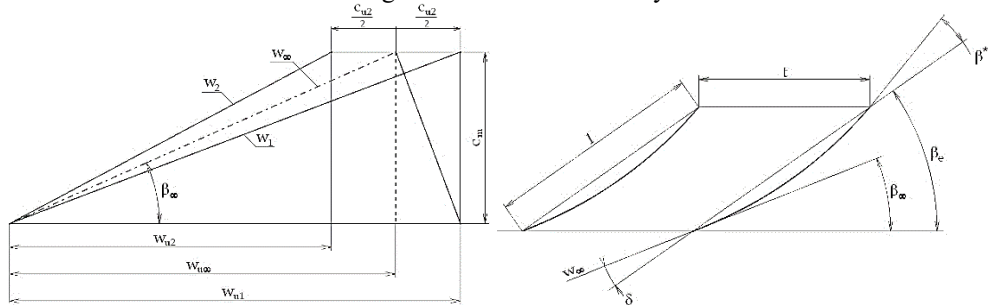


Fig. 6. Velocity triangles and blade cascade of axial pump runner [3].

The procedure is as follows. The peripheral component of the mean relative velocity $w_{u\infty}$ (6), angle β_{∞} (7) and angle of the cascade's inclination β_e (8) are determined, while the angle of attack δ is considered in the zero approach equal to zero. Parameter η_H in equation (6) is the hydraulic efficiency, which is determined on the basis of diagrams compiled according

to experimental data. Next, the length of the blade chord (l) is calculated according (9) and relative pitch as well (10).

Coefficient of cascade's impact ξ , can be calculated from (11), but it is possible to determine it by the diagram in the Fig. 7.

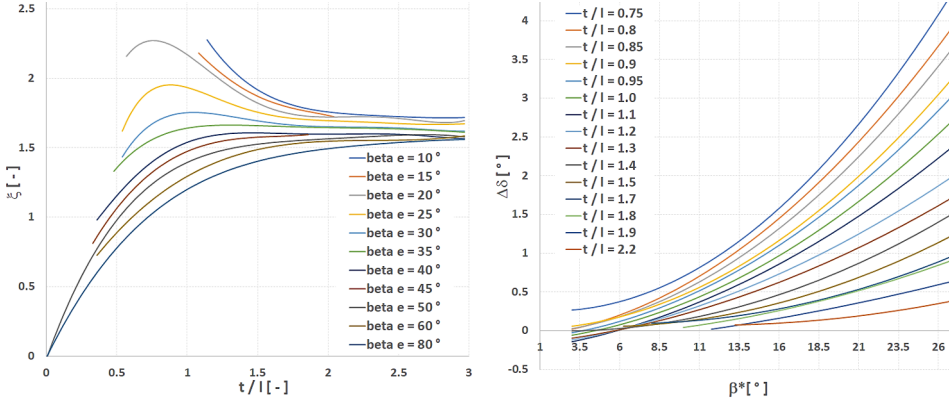


Fig. 7. Dependency of the coefficient of cascade's impact ξ and changing of the angle of attack $\Delta\delta$.

Value of ξ depends on relative pitch t/l and angle of the cascade's inclination β_e (Fig. 7). It is one of the most important coefficients that determines the ratio between the lift coefficients of an isolated airfoil in a wind tunnel $(c_z)_1$ (14) and in a blade cascade c_z . We need to know this ratio because experimentally determined values (c_z) of a NACA profile were obtained on an isolated profile in a wind tunnel. Therefore, we have to recalculate the lift coefficient of the airfoil in the cascade to the value that the same airfoil would have if it were isolated. In addition, the difference in the lift coefficients of the isolated airfoil and the airfoil in the cascade, due to the fact that the airfoil in the cascade can be considered as an airfoil of infinite length, must be taken into account. This difference is compensated by changing the angle of attack of the profile $\Delta\delta$. The value of the changing of the angle of attack $\Delta\delta$ is shown in the diagram in the Fig. 7. This value depends on the relative pitch t/l and the angle of profile's deflection (camber) β^* (13) (Fig. 6). Then the mean geometric velocity (w_∞) is calculated according to equation (12).

After determining the change in the angle of attack $\Delta\delta$ a detailed calculation of $(c_z)_1$ must be performed. This calculation takes into account the value of $\Delta\delta$ in the calculation of the angle of cascade's inclination β_e (Fig. 7). This means that it is necessary to recalculate the angle of cascade's inclination according to (15) and then repeatedly perform the calculations (9) – (14).

Subsequently, it is possible to make a comparison with the experimentally determined lift coefficient (c_z) of the NACA profile, with the certain blade chord length l , relative pitch t/l , angle of attack δ and the maximum blade profile thickness t^* . Subsequently, all parameters (16) – (21) are recalculated.

A control of the lift coefficient (c_z) of the NACA profile is performed according to equation (16). The selected parameters are the relative maximum thickness of the profile t^*/l and the ratio of distance of maximum camber to chord length L^*/l . With the application of these parameters and the calculated values of ratio of maximum camber to chord length m/l and β_e it is possible to determine the geometry (coordinates) of the designed NACA airfoil. The angle of attack must be iterated to achieve an approximate match (c_z) and $(c_z)_1$.

Table 2. Equations used in the design of the blade profile.

Peripheral component of the mean relative velocity	$w_{u\infty} = u - \frac{c_{u2}}{2} = u - \frac{g \cdot H}{2 \cdot \pi \cdot D \cdot n \cdot \eta_H}$	(6)
Angle	$\beta_{\infty} = \arctg \frac{c_m}{w_{u\infty}}$	(7)
Angle of the blade cascade's inclination	$\beta_e = \beta_{\infty} + \delta$	(8)
Blade chord length	$l = \frac{\varphi_r \cdot \frac{D}{2}}{\cos \beta_e}$	(9)
Relative pitch	$\frac{t}{l} = \frac{\pi \cdot D}{z \cdot l}$	(10)
Coefficient of cascade's impact	$\xi = \frac{(c_z)_1}{(c_z)}$	(11)
Mean geometric velocity	$w_{\infty} = \sqrt{c_m^2 + w_{u\infty}^2}$	(12)
Angle of profile's deflection (camber)	$\beta^* = \frac{\frac{g \cdot H}{z \cdot n \cdot \eta_H}}{\xi \cdot w_{\infty} \cdot l}$	(13)
Lift coefficient of the isolated profile	$(c_z)_1 = \frac{\frac{2 \cdot g \cdot H}{z \cdot n \cdot \eta_H} \cdot \frac{1.6}{\xi}}{l \cdot w_{\infty}}$	(14)
Angle of the blade cascade's inclination with the changing the angle of attack	$\beta_e = \beta_{\infty} + \delta + \Delta \delta$	(15)

Table 3. Equations used for determining of the lift coefficient of the NACA profile.

Lift coefficient on the NACA profile	$(c_z) = \frac{\partial c_z}{\partial \delta} \cdot (\delta_{\lambda} - \delta_0)$	(16)
Ratio	$\frac{\partial c_z}{\partial \delta} = 0.079 - 0.037 \cdot \frac{t^*}{l}$	(17)
Angle of attack by zero lift	$\delta_0^{\circ} = -83.3 \cdot \frac{m}{l} \cdot \left(0.74 + \frac{L^*}{l}\right)$	(18)
Ratio of maximum camber to chord length hrany (m/l) _{40%} (distance $L^* = 0.4 \cdot l$)	$\frac{m}{l} = \frac{\sin^2 \left(\frac{\beta^*}{2}\right)}{\sin \beta^*} - \frac{1}{20} \cdot \tg \frac{\beta^*}{5}$	(19)
The angle of attack for the final span	$\delta_{\lambda} = \delta_i + \Delta \delta$	(20)
The induced angle of attack	$\delta_i = 57.3 \cdot \frac{(c_z)_1}{\pi \cdot \lambda}$	(21)

Calculated parameter values l , β_e , $\frac{m}{l}$, t^* are used to calculate the coordinates of points of specific blade cuts – blade profiles on streamline surfaces "B – A" marked in Fig. 3. The calculation is carried out on the basis of equations describing the relative values of the coordinates of a specific NACA profile.

The mentioned procedure was implemented in our own "in – house" calculation software (created in the Excel VBA environment). This ensured the automatic design of the runner geometry. The designed geometry of the runner is subsequently exported to the simulation software environment via text files containing the coordinates of the blade cuts (profiles).

3 CFD simulation of a turbine with a new runner

CFD simulation was performed on a full turbine flow domain. Each simulated operating mode took into account the specific position of the runner blades and the distributor vanes.

The computational mesh was purely hexahedral with a total number of cells of approximately 7 million (Fig. 8).

A steady – state, incompressible, isothermal flow model was applied, including the solution of the Navier – Stokes equations, the continuity equation and the k- ϵ RNG turbulence model equations. "Mixing – plane" interfaces between stationary and rotating subdomains were implemented.

The 2nd order discretization schemes were applied. Boundary conditions: total pressure and intensity of turbulence – inlet, static pressure – outlet, rotating speed.

CFD simulation was performed for 53 operating modes with different positions of the distributor vanes and the runner blades. The simulated operating modes included modes related to "guaranteed" head, operating range boundaries and operating points near BEP.

Integral values of capacity, torque of the entire rotor, power, efficiency were evaluated. The comparison of the required and calculated parameters (of the turbine with the new runner – new design) for the guaranteed head is shown graphically in the Fig. 9. CFD simulations were performed identically on the original turbine (with the original runner). The Fig. 10 shows the calculated parameters of the original turbine and calculated parameters of the turbine with the new runner. The difference in parameters between the original and modified turbine (new design) is obvious.

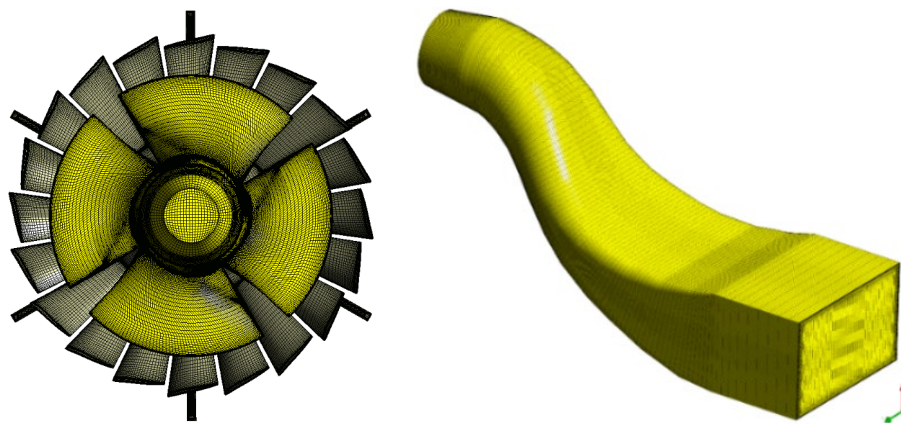


Fig. 8. Computational mesh for CFD simulation.

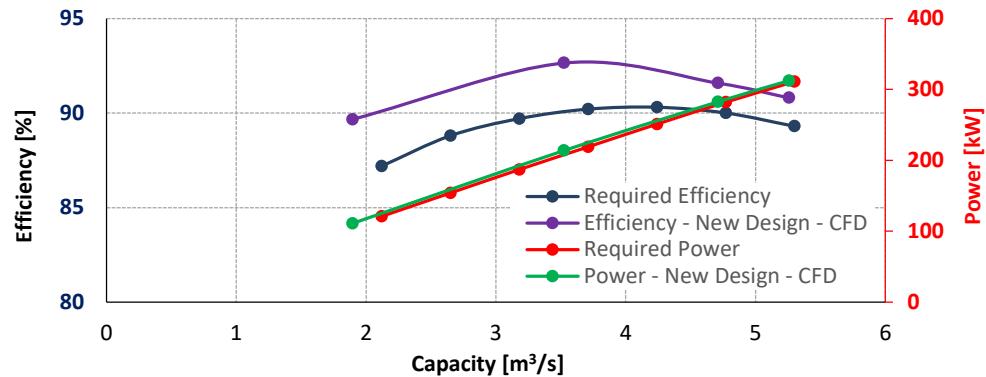


Fig. 9. Comparison of required (contracted) efficiency and power values (blue or red curve) with calculated (CFD) efficiency and power values – new design (purple or green curve), $H = 6.7$ m.

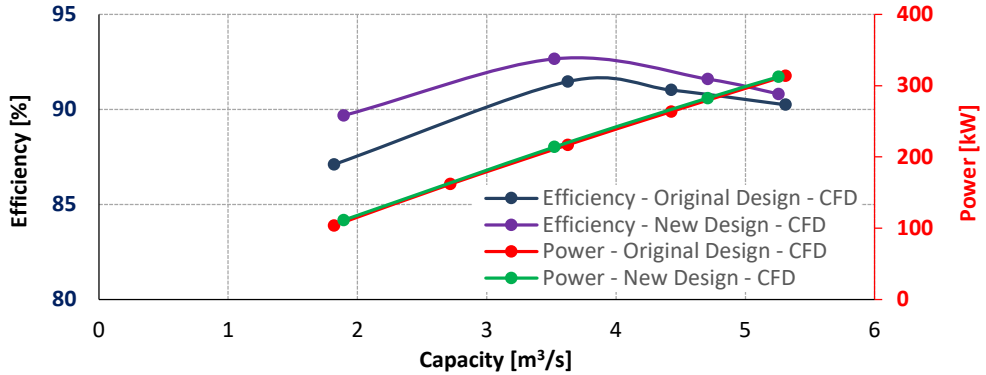


Fig. 10. Comparison of calculated (CFD) efficiency and power values – original design (blue or red curve) with calculated efficiency and power values – new design (purple or green curve), $H = 6.7$ m.

4 Validation of CFD simulation

Results of model tests performed on the original turbine model were compared to the results of CFD simulation of the turbine with original geometry with model size (Fig. 11).

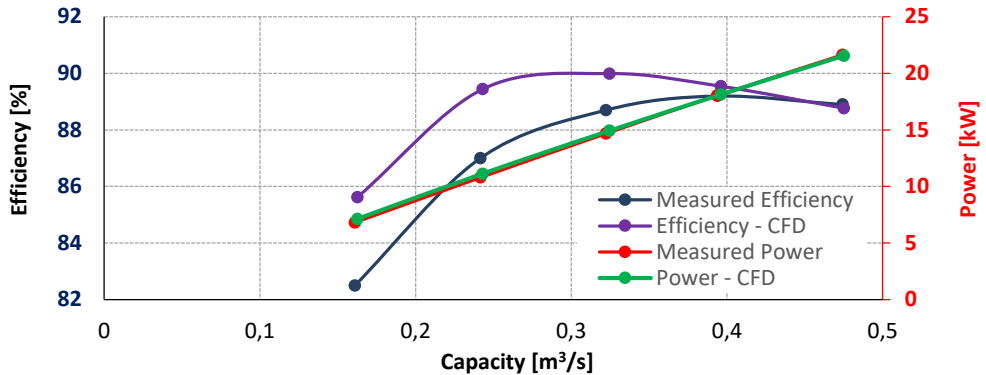


Fig. 11. Comparison of measured efficiency and power values – model test (blue or red curve) with calculated efficiency and power values (purple or green curve) – original geometry, $H = 5.23$ m.

It can be seen from the Fig. 11 that a relatively good agreement is achieved between the results of the measurements and the CFD simulation.

5 Conclusion

A CFD simulation showed that the runner designed using the classical method meets all the required parameters. It should be remarked that the new runner geometry is still the primary geometry, which could be optimized using one of the approaches stated in the introduction of this paper. It turned out that the hydraulic design carried out by the classical approach with the verification of parameters by CFD simulation is a practical and time-saving way to fulfill the assignment. The advantage of the classical approach to the design of a primary geometry is its simplicity and the explicit design of the geometry of the runner based

on the required parameters at the BEP. Subsequent more complex optimization algorithms ensure automatic optimization of the hydraulic shape of the runner, while a greater number of design variants are analyzed. Automatic optimization can provide slightly better parameters compared to the "engineering" approach to a runner optimization, but at the cost of increased computing time while the achieved benefit may not always be sufficient.

Authors would like to thank to project: Regenerácia použitých batérií z elektromobilov / Regeneration of used batteries from electric vehicles ITMS2014+: 313012BUN5

References

1. Varchola, M., Hlbočan, P. (2016). *Hydraulický návrh axiálneho stroja. 1. vyd.* STU v Bratislave.
2. Hlbočan, P., Varchola, M. (2015). CFD modelovanie v procese návrhu axiálneho stroja. In *ESPM 2015. Technické vedy a výrobný manažment 2015 : zborník abstraktov z medzinárodnej vedeckej konferencie*. S. 49.
3. Varchola, M., Bielik, T., Hlbočan, P. (2013). Methodology of 3D hydraulic design of a impeller of axial turbo machine. In *Engineering Mechanics. Vol. 20, No. 2, s.107-118*.
4. Hlbočan, P., Varchola, M. (2016). CFD simulation in process of axial turbo machine design. In *Production Management and Engineering Sciences. CRC Press. Taylor & Francis Group*, S. 383-389.
5. Hlbočan, P., Varchola, M. (2013) Numerical simulation on a mixed-flow pump operating in a turbine mode. In *Engineering Mechanics. Vol. 20, No. 2, s.97-105*.
6. Hailu, G., Varchola, M., Hlbočan, P. (2011). *Design of Hydrodynamic Machines Pumps and Hydro-Turbines*. CRC Press. Taylor & Francis Group, 266 Pages.
7. Bielik, T. (2010). *Hydraulický návrh axiálneho stroja v 3D prostredí*, Diplomová práca SjF STU v Bratislave.
8. Vrábel, M. (2014). *Hydraulický návrh axiálneho stroja s výstupom pre CFD softvér*, Diplomová práca SjF STU v Bratislave.
9. Strýček, O. (1994). *Hydrodynamické čerpadlá*, STU v Bratislave
10. Brůha, O., Felber, V. Smolař, V. (1939). *Letecký Průvodce Díl 2*, Matice česká.
11. Nechleba, M. (1962). *Vodní Turbíny jejich konstrukce a příslušenství*, SNTL Praha.
12. Lewis, R., I. (1996) *Turbomachinery performance analysis*, Elsevier Science and Technology books.
13. Lobanof, V., S., Ross, R., R. (2013). *Centrifugal Pumps – Design and Application*. Gulf Professional Publishing.
14. Kantor, M., Chalupa, M., Souček, J., Bílková, E., Nowak, P. (2010). Application of Genetic Algorithm Methods for Water Turbine Blade Shape Optimization. *Manufacturing Technology, Vol. 20, No. 4*.
15. Semenova, A. et al. (2014). Multi-objective shape optimization of runner blade for Kaplan turbine, *IOP Conf. Ser.: Earth Environ. Sci.* 22 012025.