Design of a pump-turbine using a quasi-potential flow approach, mathematical optimization and CFD

F Pochyly¹, P Rudolf², D Štefán¹, P Moravec¹, J Stejskal¹ and A Skoták²

¹ V. Kaplan Department of Fluid Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2896/2, 61699 Brno, Czech Republic
² ČKD Blansko Engineering, a.s., Čapkova 2357/5, 678 01 Blansko, Czech Republic
E-mail: stefan@fme.vutbr.cz

Abstract. Design of a pump turbine \( n_s = 135 \) is carried out using a very fast quasi-potential flow approach for the preliminary design. Theory behind this novel method is presented and its advantages discussed. Further finetuning is performed by coupling of CFD (ANSYS CFX) and particle swarm optimization method. The final design features suppression of the vortex rope over relatively wide turbine operating range. Simulations are verified by model testing in hydraulic laboratory.

1. Introduction

The new blade profile for the pump turbine runner was developed using a combination of parametrized rotational flow model [1], [2] and particle swarm optimization algorithm (PSOA) [3]. The design goal was a refurbishment of an old pump turbine runner, while keeping the rest of geometry (i.e. spiral case, guide vanes and the draft tube). The main priorities were broad operating range, thus prevention of vortex rope and rotating stalls formation and cavitation resistance [4],[5],[6]. Theory of quasi-potential flow approach was built using curvilinear coordinates and compound lamellar flow (CLF), also called a quasi-potential flow. [7]. The CLF is derived to be compromise between potential flow (low velocity near the hub, thus very small blade inlet angle resulting into low suction ability) and Francis method (assumption of constant velocity profile, thus far from the reality). The meridional velocity profiles resulting from potential flow approach \( v_p \) and Francis method \( v_f \) are schematically shown in Figure 1. Using the compound lamellar flow approach (quasi-potential method), the meridional flow is set to be more or less close to one of these methods. This choice enables to influence progression of the blade angle \( \beta \) which is one of the key factors influencing the performance of the runner [8], [9]. The shape of the blade itself is then calculated considering the potential flow.

2. CLF theory

Using the curvilinear coordinates it is possible to easily construct curves and surfaces in the space. The Bezier surface described by equation (1) is geometrically very flexible and mathematically convenient tool.
The curvilinear coordinates \( u^1, u^2 \) are chosen so that coordinate curves create orthogonal grid. The third coordinate represents the wrap angle \( u^3 = \phi \). In such case the tangential vectors \( g_i \), depicted in Figure 2, are perpendicular to each other. The shape of curve, respectively surface, is determined by choosing polygon’s points \( b_{i,j} \) to determine the orthogonal grid. The \( B_i \) and \( B_j \) are Bernstein polynomials. The direction of \( u^2 \) corresponds to streamline of meridional velocity \( v_m \). Under this assumption the Euler turbomachinery equation is modified and the formulas for specific energy \( Y \) are derived in form of equations (2 - 3), where \( F \equiv v_m g_2, \, \Omega = 2\pi n \) is angular velocity, \( \alpha = \frac{g_1 r g_2}{u^1}, \, \gamma = \frac{g_1 g_2}{u^1 r^2} \) and \( \lambda \) is the vorticity contribution (\( \lambda = 0 \) for potential flow, otherwise the flow is rotational). For more details the reader is referred to [1], [2].

\[
\frac{1}{\sqrt{g}} \frac{\partial F}{\partial u^1} = \lambda \frac{r}{r^2} \left( Y - \frac{\Omega r^2}{2} \right) \tag{2}
\]

\[
\frac{\partial}{\partial u^2} (\alpha F) + \gamma \frac{\partial Y}{\partial u^3} = 0 \tag{3}
\]

**Figure 1.** Velocity profiles of potential flow \( v_p \) (dash-dotted line) and Francis method \( v_f \) (dashed line)

**Figure 2.** Orthogonal curvilinear coordinates in the meridional cross section.

For the basic blade design it is necessary to consider the flow without spiral vortices. This assumption is fulfilled by special form of rotational flow, so called quasi-potential flow, according to condition (4).

\[
\text{rot} \, \mathbf{v} \cdot \mathbf{v} = 0 \tag{4}
\]

The function \( \mathbf{v} = \kappa \, \text{grad} \Phi \) satisfies the equation (4). The meridional flow is calculated considering equation (3) and flow kinematics must fulfill the continuity equation which is interpreted in curvilinear coordinates by equation (5).

\[
\frac{g_1(u^1,1)r(u^1,1)g_2(u^1,u^2)}{g_1(u^1,u^2)r(u^1,u^2)} v_{m2} = \frac{\kappa(u^1,u^2)Q}{2\pi \int_0^1 \kappa(u^1,u^2)g_1(u^1,u^2)r(u^1,u^2)du^1} \tag{5}
\]
Based on selection of $\kappa$ values, it is possible to obtain different velocity profiles of $v_m$, for example:

(i) Francis method $v_m = A(u^2)$, $\kappa(u^1, u^2) = g_2$
(ii) Irrotational flow $\text{rot} v_m = 0$, $\kappa(u^1, u^2) = 1$
(iii) Rotational flow (CLF) $\text{rot}(\frac{1}{n} v_m) = 0$, $\kappa = g_n^2$, where $n \in [0, 1]$

3. Runner design

By using our in-house software the initial blade profile was calculated for the pump mode, i.e. axial to radial flow in meridian section. The new meridional profile of the runner was constrained only by the distributor (guide vanes) and draft tube geometry. In other words, the dimensions $D_1$, $D_2$, $B$ and $H$, shown in Figure 3, remained unchanged. The spiral casing contains 12 stay vanes (including the tongue) and the distributor and runner have 24 guide vanes and 9 blades respectively.

![Figure 3. Meridional cross-section.](image)

At the beginning the meridional profile is built using Bezier surface controlling the evolution of flow cross-section, see Figure 4. The flow cross-section should gradually increase from inlet to the outlet of impeller, without large variations. The blue points are controlling points of Bezier surface. The meridional shape is interactively modeled by movements of controlling points. Consequently, the shape of newly created hub and shroud was modified as shown in Figure 5, where the initial design is the meridional profile of old runner and new design is the meridional profile created using CLF. In next step the meridional velocity $v_m$ for design parameters $(Q, H, n)$ is calculated as shown in Figure 6. All dimensions are related to the reference radius $R_{ref}$ corresponding to the runner-draft tube interface and $v_m$ is made dimensionless according to reference velocity $v_{ref} = \frac{Q}{\pi R_{ref}}$. According to runner angular speed, chosen blade angle at trailing edge and previously calculated meridional flow, the blade shape is calculated, see Figure 7. This is the final step of CLF design procedure itself and this blade profile is further optimized using PSOA as described in following section.
4. Optimization process
Using the Particle Swarm Optimization Algorithm (PSOA) only the blade angles were optimized keeping the fixed meridional geometry and the main runner dimensions [10], [11]. The optimization cycle consists of several steps:

(i) Main optimization procedure – Particle Swarm Optimization
(ii) Geometry set-up in BladeGen software
(iii) Mesh creation in TurboGrid software
(iv) Numerical simulation in ANSYS CFX software
Table 1. Overview of PSOA parameters.

| Parameter                  | Value |
|----------------------------|-------|
| Number of particles        | 5     |
| Inertia weight             | 0.8   |
| PSO parameters             | c1, c2| 2     |
| Maximum velocity           | $v_{max}$ | 0.01 |

4.1. Particle Swarm Optimization

Particle Swarm Optimization algorithm [3] (shortly PSOA) was utilized for shape optimization of the pump turbine impeller mainly as its Global Best modification. PSOA is a global optimization method with stochastic nature, but faster than genetic algorithm. This method uses swarm of particles randomly distributed in given computational area (in our case constrained by a maximal and minimal value of blade $\beta$ angle). These particles move in such domain according to equations [12]:

$$v_{in} = w \cdot v_{in} + c_1 \cdot \text{rand()} \cdot (p_{in} - x_{in}) + c_1 \cdot \text{Rand()} \cdot (p_{gn} - x_{in})$$  (6)

where $n$ – problem dimension; $v_{in}$ – velocity of $i$-th particle (step size); $w$ – inertia weight; $c_1, c_2$ – PSO parameters; $p_{gn}$ – position of the particle with best value of examined function (Global Best). Variables rand() and Rand() are vectors (they are not equal), containing random numbers from interval (0,1). These random numbers endorse the stochastic behaviour of PSOA. And as defined in [12], the particle position $x_{in}$ is calculated:

$$x_{in} = x_{in} + v_{in}$$  (7)

where $x_{in}$ – position of $i$-th particle; $v_{in}$ – velocity of $i$-th particle (step size). Each individual particle carries information about specific design, thus each sample of the initial swarm corresponds to a given (random) set of geometry parameters. Main idea of PSOA is that all particles converge to one (same) solution (global optimum), in our case all particles grouped in an area of similar values of pump $H$ and $\eta_h$ and have similar sets of geometry parameters. In equation (6) and PSOA process several parameters appear, which radically influence behavior of the computational swarm. Parameters are listed in Tab. 1.

4.2. Examined function

Main goal of PSOA algorithm was set to minimize function defined as:

$$f(x) = weight_1 \cdot |1 - (H_{CFD} / H)| + weight_2 \cdot |1 - \eta_{H,CFD}|$$  (8)

where $weight_1, weight_2$ [-] – objective function optimization weights (chosen value = 1); $H_{CFD}$ (m) – pump turbine delivery height from CFD; $\eta_{H,CFD}$ (-) – hydraulic efficiency from CFD; $H$ (m) – pump delivery height characterized by an average value of maximum and minimum requested value of a pump turbine delivery height.

4.3. Pump turbine delivery height definition

$$H_{CFD} = \frac{p_{total, out} - p_{total, in}}{\rho g}$$  (9)

where $p_{total, out}$ (Pa) is total pressure at computational outlet, $p_{total, in}$ (Pa) is total pressure at computational inlet.
Table 2. Performance comparison of initial and optimized impeller design states.

|                  | $H_{CFD}/H_{DesignCFD}$ | $\eta_{H,CFD}/\eta_{H,DesignCFD}$ |
|------------------|--------------------------|-----------------------------------|
| Initial state (CLF) | 0.876                    | 0.965                             |
| Final state (Design)(PSO) | 1 +12.4%                | 1 +3.5%                           |

4.4. Hydraulic efficiency definition

$$\eta_{H,CFD} = \frac{(p_{total \_out} - p_{total \_in})Q}{2\pi n M_k}$$

(10)

where $n$ ($s^{-1}$) is runner rotational speed and $M_k$ ($Nm$) is torque.

4.5. Impeller model

Every impeller model was created and parameterized in commercial software called BladeGen. BladeGen was controlled via text file *.bgi, which contains information about main parameters of meridional flow channel and blade (e.g. $\beta$-angle, blade thickness etc.). Parametrization takes place in 6 locations along three streamlines (near hub, near shroud and middle streamline) resulting into 18 modified parameters of blade $\beta$-angle. Restriction bounds for each streamline had same values and were determined around a development of $\beta$-angle from CLF design. The bounds magnitude varies form leading edge (the smallest bounds) to trailing edge (the largest bounds) and its determination was based on a “trial and error” method. The bounds must be as small as possible to suppress wide spread of final results (smaller bounds = smaller computational area for the swarm).

4.6. CFD simulation of optimization loop

Computational mesh of the pump-turbine impeller was created in commercial software called TurboGrid. As the input for this software served curves of hub, shroud and blades of BladeGen main output. Mesh was generated as one periodical flow channel with hexahedral elements with average count 450000. CFD simulations within optimization loop were carried out by ANSYS CFX software. The steady state flow with one periodic impeller channel and with stage (mixing plane) interface was chosen (mainly for computational time savings). The standard $k-\epsilon$ turbulence model was employed and limitation of 5000 solver iterations was imposed. The pump-turbine was running in pump mode, thus the inlet boundary condition in draft tube was set as mass flow $Q_m$, which corresponded to the average between maximum and minimum given flow rate. Outlet boundary condition in spiral casing was set as zero relative static pressure $p = 0 \text{ Pa}$. The distributor opening was selected to be average value between maximum possible opening and fully closed guide vanes.

4.7. Results of optimization

Optimization process was stopped by the user after six iterations (6. gen.), when all particles grouped into near surrounding as shown in Figure 8. This figure also captures progressive movement of the swarm. Pump delivery height $H_{CFD}$ and hydraulic efficiency $\eta_{H,CFD}$ were computed from averaging last 500 iterations based on equations (9) and (10). The final shape of the impeller was chosen from the last generation of the swarm based on equation (8). Both initial and optimized design states are compared in terms of efficiency and delivery height in Figure 9 and summarized in Table 2.
5. Experimental measurements
The newly designed runner (model size with runner diameter $D_m = D_P / 6.4844$) was measured in the hydraulic laboratory of our industrial partner ČKD Blansko Engineering. Since the blade design procedure and consequently the shape optimization were based on prototype dimensions and parameters, the measured data of head $H_p$ and flow rate $Q_p$ are converted to the prototype size using following equations:

$$H_p = \frac{H_m \cdot (n_p \cdot D_p)^2}{(n_m \cdot D_m)^2} \tag{11}$$

$$Q_p = Q_m \cdot \left(\frac{D_p}{D_m}\right)^2 \cdot \left(\frac{H_p}{H_m}\right)^{0.5} \tag{12}$$

Both the pump mode and turbine mode were measured considering several guide vane openings $a_0$. The $Q - H$ and $Q - \eta$ characteristics of pump mode are shown in Figure 10 and Figure 11 respectively. All quantities are made dimensionless with respect to the maximum efficiency. The design point of requested $H$ for specified flow rate $Q$ is shown as black cross. It is noticeable that the design parameters were successfully achieved both in terms of the required delivery height and the position of maximum efficiency.

6. Correlation of computed and measured data
For purpose of data correlation between measurements and CFD simulations the unsteady (URANS) simulations were done. The grids of all turbine domains (spiral casing, distributor, runner and draft tube) were built in ICEM as fully hexahedral consisting of 12 mil. elements, see Figure 13. The CFD calculations were done in Ansys CFX utilizing the standard $k - \epsilon$ turbulence model with scalable wall function. The “High Resolution” option was selected for the advection scheme, ”Second Order Backward” for the transient scheme and ”First Order” for turbulence numerics. The time-step size corresponds to 1° of runner revolution.

From the unsteady simulations, the turbine hill chart was built to identify maximum efficiency $\eta$. The efficiency for both measured and simulated hill charts was made relative to the maximum value of each data set. The computed maximum efficiency was approximately 2.6% higher than the measured one. One of the reasons is that the leakage flow through the labyrinths seals was not modeled in CFD which would decrease computed efficiency by more than $1 - 2\%$ at best efficiency point [13], [14]. The hill chart comparison is plotted in Figure 14 where the blue cross...
marks the nominal parameters for turbine mode. The coordinates of unit speed $n_{11}$ and unit flow rate $Q_{11}$ are defined in equation (13).

$$Q_{11} = \frac{Q}{D^2 \sqrt{H}}, \quad n_{11} = \frac{nD}{\sqrt{H}}$$

One can see that for turbine mode the maximum efficiency is achieved for lower $n_{11}$ and $Q_{11}$ than the design point parameters. The reason is that the pump turbine was optimized only for the pump mode.

The comparison of pump mode for one selected guide vane opening $a_0 = 129.69$ mm is shown in Figures 15 - 17. While the curve slope of measured delivery height is predicted well for higher flow rates $Q/Q_{opt} > 0.7$, the underestimation is observed for range $0.35 < Q/Q_{opt} < 0.7$ and large overestimation for $Q/Q_{opt} < 0.35$ (partial load). Similarly the simulated torque is largely overestimated for $Q/Q_{opt} < 0.35$. The exact cause of this behavior is going to be investigated in future, but from preliminary post-processing of CFD results the main source might be the flow complexity, such as rotating stalls in the runner and reverse swirling flow in distributor and spiral casing. Specifically at $Q/Q_{opt} = 0.526$ the large vortex structures develop between the stay vanes causing the reversal flow back towards the guide vanes (see Figure 18). Such swirling
Figure 14. Hill chart of normalized efficiency, measured in black and simulated in red, blue cross is design point of turbine operations.

Figure 15. Comparison of measured and computed delivery height.

Figure 16. Comparison of measured and computed hydraulic efficiency.

flow is generally difficult to simulate correctly employing two-equation RANS turbulence model. This can cause inaccurate solutions for strong streamlines curvature flows and flows tending to separate.

7. Conclusions
Using the design procedure combining the quasi-potential flow approach, mathematical optimization and CFD simulation we presented fast and convenient tool to built new runner geometry of pump-turbine runner. Surprisingly, the impeller design provided by CLF method (i.e. quasi-potential flow) did not need too much of further finetuning using coupled PSOA and CFD technique. The measurements of pump-turbine model carried out in hydraulic laboratory revealed that presented design methodology is efficient both in terms of parameters fulfillment
and usability in wide operating range. In future the coupled optimization of pump and turbine mode must be considered in order to fulfill required parameters also in turbine mode.

Acknowledgments
This work is an output of cooperation between Technology agency of the Czech Republic project TH01020982 and NETME Centre, regional R&D centre built with the financial support from the Operational Programme Research and Development for Innovations within the project NETME Centre (New Technologies for Mechanical Engineering), Reg. No. CZ.1.05/2.1.00/01.0002 and, in the follow-up sustainability stage, supported through NETME CENTRE PLUS (LO1202) by financial means from the Ministry of Education, Youth and Sports under the "National Sustainability Programme I".

References
[1] Pochyly F and Stejskal J 2016 J. Fluids Eng 138
[2] Stejskal J 2017 Analysis of the velocity and pressure fields of the liquid using curvilinear coordinates Ph.D. thesis Brno University of Technology
[3] Kennedy J and Eberhart R 1995 The IEEE International Conference on Neural Networks vol 4 pp 1942–1948
[4] Skotáč A, Feilhauer M, Obrovský J, Stégner L, Veseľý J and Čepa Z 2015 2015 16th International Scientific Conference on Electric Power Engineering (EPE) (Kouty nad Desnou, Czech Republic: IEEE) pp 184–187
[5] Braun O, Kueny J L and Avellan F 2005 ASME 2005 Fluids Engineering Division Summer Meeting vol 1 (Houston, Texas, USA) p 6
[6] Edinger G, Erne S, Doujak E and Bauer C 2014 IOP Conference Series: Earth and Environmental Science 22 032051
[7] Smirnoff V I 1964 A Course of Higher Mathematics vol 2 (Oxford, UK: Pergamon Press)
[8] Gălib J F 2010 Centripetal Pumps vol 2 (Springer, Berlin, Heidelberg)
[9] Song P, Zhang Y, Xu C, Zhou X and Zhang J 2015 International Journal of Fluid Machinery and Systems 8 94–101
[10] Moravec P and Rudolf P 2017 EPJ Web Conf. 143
[11] Moravec P and Rudolf P 2018 Advances in Hydroinformatics 1 ed Gourbesville P, Cunge J and Caignaert G (Springer Singapore) pp 997–1012
[12] Shi Y H and Eberhart R 1998 The Int. Conf. on Evolutionary Computation pp 69–73
[13] Schiffer J, Benigni H and Jaberg H 2017 J. Fluids Eng 139
[14] Celič D and Ondráčka H 2015 Journal of Physics: Conference Series 579