Modifications of propeller pumps design algorithm. 
Numerical and laboratory tests

I Altyntsev\textsuperscript{2}, D Homa\textsuperscript{2}, G Peczkis\textsuperscript{1} 

\textsuperscript{1}Institute of Power Engineering and Turbomachinery, Silesian University of Technology, Poland 
\textsuperscript{2}POWEN-Wafapomp Group, Zabrze, Poland 

E-mail: peczkis@wp.pl

Abstract. This article presents the new construction of a centrifugal impeller pump which supplies the cooling system in coal power plants. The article also shows the history of producing this kind of pumps for unified power units of 125 MW and 200 MW which dominate in Poland. The process of numerical analysis and conventional calculations leading to the improvement of qualities of the flowing system exceeding literature indicators are presented. The article shows the problems which appeared during assembling the pump in its target workplace. The results of the measurements of the pump flowing system in its target workplace are presented.

1. Introduction
Propeller pumps are widely used in cases when high flow rate is needed with a relatively low head. Propeller pumps capacities range from several to several dozens of cubic meters per second, with head from 4 – 12 m \cite{1,2}. Specific speed of propeller pumps varies from 135 – 600.

The algorithm to determine the pump geometry is different for propeller pumps than for centrifugal pumps. In centrifugal pumps the flow is analyzed as a flow through the inter-vanes channels. In propeller pumps, the blades form a cylindrical palisade, the inter-blade ducts are very short and theory of inter-blades flow is replaced by the analysis of flow over the foils in palisade \cite{3}.

The two most popular design methods based on aerodynamic theory are \cite{4}:
- Bauersfeld, Wozniesienski method - involves analysis of the flow around an insulated foil, and then makes adjustments due to the location of the foil in the cylindrical palisade 
- Martensen, Riegels method - involves analysis of the flow through a cylindrical palisade of blades with a specific profile and density.

The purpose of this article is to present the Wozniesienski method with some modifications that the Authors propose based on their own experience with the design of propeller pumps.

2. Algorithm to determine rotor design parameters
When starting the calculations, the basic operating parameters of the pump should be determined, such as head \(H\), flow rate \(Q\) and rotational speed \(n\). Firstly, it is necessary to check the value of the specific speed in accordance with the relationship:
The rest of the algorithm uses the specific speed calculated on the basis of specific work $Y$

$$Y = gH$$  \hspace{1cm} (2)

$$n_b = n \frac{\sqrt{Q}}{Y^4}$$  \hspace{1cm} (3)

$$n_q = 1213.9 n_b$$  \hspace{1cm} (4)

The total efficiency can be determined from one of the dependencies [2]:

$$0.04 \leq n_b \leq 0.33$$

$$\eta_c = \sqrt{\left( \frac{Q}{0.048} \right)^{0.083} - | -0.722 - \log n_b |^3}$$  \hspace{1cm} (5.1)

$$n_b > 0.33$$

$$\eta_c = \sqrt{\left( \frac{Q}{0.048} \right)^{0.07} - | -0.722 - \log n_b |^5}$$  \hspace{1cm} (5.2)

$$Q > 0.65 \text{ m}^3/\text{s}$$

$$0.04 \leq n_b \leq 0.33$$

$$\eta_c = \sqrt{1.24 - | -0.722 - \log n_b |^3 - 0.2}$$  \hspace{1cm} (5.3)

$$n_b > 0.33$$

$$\eta_c = \sqrt{1.24 - | -0.722 - \log n_b |^5 - 0.2}$$  \hspace{1cm} (5.4)

The hydraulic efficiency (Wislicenus formula):

$$\eta_h = \sqrt{\eta_c - (0.02 - 0.06)}$$  \hspace{1cm} (6)

The formulas for determining the overall efficiency and hydraulic efficiency are only approximations, the actual value of efficiency may differ from them.

Mechanical power (on the shaft):

$$p = \frac{\rho Q g H}{\eta_h}$$  \hspace{1cm} (7)

The next step is to determine the meridional velocity $c_m$:

$$c_m = K_{mc} \sqrt{2 g H}$$  \hspace{1cm} (8)

Coefficient $K_{mc}$ is calculated as follows:

$$K_{mc} = 0.0688 + 0.733 n_b^{1.1}$$  \hspace{1cm} (9)

Meridional velocity at the rotor inlet:
where \( Q_v \) - volumetric flow losses, \( D_B \) – rotor outer diameter, \( D_A \) – hub diameter. Volumetric flow losses are assumed in the range of 2% - 4% of nominal flow rate. The dependency between \( D_A \) and \( D_B \):

\[
\frac{D_A}{D_B} = 0.63 - 0.346 \cdot (n_b - 0.25)
\]

After transformation:

\[
D_B = \sqrt{\frac{4(Q + Q_v)}{\pi}} \cdot \frac{1}{c_m} \cdot \left(1 - \left(\frac{D_A}{D_B}\right)^2\right)^2
\]

**Figure 1. The subsequent rotor streamlines.**

In some cases, when the set head is significant and the specific speed indicator oscillates on the border of the propeller and diagonal pump, it is advisable to abandon the constant value of \( D_A \) diameter in favor of increasing \( D_{A2} > D_{A1} \) diameter on the flow path. In this case, an intermediate value should be used for further calculations.

The diameters of intermediate streamlines are determined based on the principle of dividing the stream into equal parts.

The next step is to determine the number of rotor blades. As the specific speed increases number of blades decreases and the maximum head achievable [5] decreases. Indicative guidelines can be found in the literature [6]. An alternative is to use the wrap angle relationship, which is described in [7]. The blade wrap angle \( \phi_r \) for the assumed number of blades should take the value between maximum \( \phi_{\text{max}} \) and minimum \( \phi_{\text{min}} \). The maximum angle is calculated as:

\[
\phi_{\text{max}} = 9.481n_B^2 - 36.426n_B + 94.502
\]

Whereas the minimum angle was assumed at the level \( \phi_{\text{min}} = 40^\circ \) (Fig. 2) [7]. Then the blade wrap angle \( \phi_r \) is calculated for the assumed number of blades. For a different number of blades, the formula defining the blade wrap angle takes different forms.

\[
Z=2 \quad \phi_r = -40.5402n_b + 121.0804 \quad (14a)
\]

\[
Z=3 \quad \phi_r = -54.589 \ln n_b + 66.96 \quad (14b)
\]
After determining the blade wrap angle for the assumed number of blades \( Z \) the following condition has to be fulfilled:

\[ \varphi_{min} < \varphi_r < \varphi_{max} \]

Next the circulation for the rotor is:

\[ \Gamma_z = \frac{gH}{\eta h \beta_0} \]  

And for one blade:

\[ \Gamma = \frac{\Gamma_z}{Z} \]  

The circulation value should theoretically cause fluid to move around the foil and create the required head \( H_{th} \). However, prof. Troskolański recommended to increase the circulation by about 10\% due to the viscosity of the liquid. In the case of significant heads, the authors propose to increase the circulation value by up to 25\%. It is reasonable to include in the calculations the power shortage factor \( k \). The value of speed circulation including the power shortage factor \( k \):

\[ \Gamma'_z = \frac{gH(1 + k)}{\eta h \beta_0} \]

where \( k \) can be assumed from 0 (no deficiency) to about 0.3.

After calculating the main parameters, you can proceed to plot blade profiles (for subsequent streamlines). The basic geometrical parameters of the profile are (Fig. 3):  
1 - chord length;

\[ \begin{align*}
Z=4 & \quad \varphi_r = -57.702 \ln n_b + 46.4 \\
Z=5 & \quad \varphi_r = -58.475 \ln n_b + 29.032 \\
Z=6 & \quad \varphi_r = -49.291 \ln n_b + 23.106
\end{align*} \]
m – maximum camber (maximum distance between the camber line and the chord line);
t* – maximum thickness of the profile;
L* – the length of the chord at which the profile has the greatest thickness.

**Figure 3.** Main dimensions of an airfoil profile.

To characterize the profile, relative values are used, referred to the length of the chord of the profile, i.e.: \( \frac{m}{T}, \frac{L^*}{T}, \frac{t^*}{T} \). Relative values are included in the profile series designation. The article presents the results for calculations using the four-digit NACA profile. The relative coordinates of the chord profile \( x \) and \( y \), are defined as:

\[
\begin{align*}
\text{from } x = 0 \text{ to } x = \frac{L^*}{l} & \quad y_s = \frac{m}{T} \left[ 2 \frac{L^*}{l} x - x^2 \right] \\
\text{from } x = \frac{L^*}{l} \text{ to } x = 1 & \quad y_s = \frac{m}{T} \left( \frac{L^*}{l} \right)^2 \left[ 1 - 2 \frac{L^*}{l} + 2 \frac{L^*}{l} x - x^2 \right]
\end{align*}
\]

**Figure 4.** The outline of the profile.

For simplicity, the NACA profile is replaced by an equivalent arc. The values of \( L^*/l \) and \( t^* \) are set, while \( l \) and \( m/l \) are calculated values. The value of \( m/l \) is calculated on basis of blade curvature angle \( \beta^* \):

\[
\left( \frac{m}{T} \right)_{40\%} = \frac{\sin^2 \left( \frac{\beta^*}{2} \right)}{\sin \beta^*} - \frac{1}{20} \tan \frac{\beta^*}{5}
\]

Half-thickness of profile \( y_t \) (relatively):

\[
y_t = \frac{t^*}{0.2} \left( 0.2969 \sqrt{x} - 0.126x - 0.3516x^2 + 0.2843x^3 - 0.1015x^4 \right)
\]

Relative coordinates of the suction (index u) and pressure (index l) sides:
\[ x_u = x - y_t \sin \theta \quad (22) \]
\[ y_u = y_z + y_t \cos \theta \quad (23) \]
\[ x_t = x + y_t \sin \theta \quad (24) \]
\[ y_t = y_z - y_t \cos \theta \quad (25) \]
\[ \theta = \tan^{-1}\left(\frac{dy_z}{dx}\right) \quad (26) \]

Radius of profile nose:
\[ r = 1.1 \left(\frac{t^*}{t}\right)^2 \quad (27) \]

The next step after plotting the profile is the slope of its chord at the appropriate angle \( \beta_e \) to the circumferential direction [8]. These calculations should be carried out for each of the designated streamlines.

\[ \text{Tangential velocity of the flow (no prerotation at inlet is assumed)}: \]
\[ \Delta c_u = c_{u2} = \frac{r_z}{2 \pi r} \quad (28) \]

Velocity \( u \):
\[ u = \frac{2 \pi nr}{60} \quad (29) \]

Relative velocity of non-disturbed flow:
\[ w_{u\infty} = u - \frac{c_{u2}}{2} \quad (30) \]
For the first iteration it is assumed that $\beta_r=\beta_\infty$, the angle of attack: $\Delta \delta=0$.  

Chord length:

$$l = \frac{\varphi_r \cdot 2\pi r}{360 \cdot \cos \beta_\infty}$$

Palisade throughput:

$$\frac{t}{l} = \frac{2\pi r}{Zl}$$

Then specify the dimensionless blade differentiator $\xi$ defined as:

$$\xi = \frac{\Gamma''}{\beta''lw_\infty}$$

It can be read from the relevant charts presented in the literature, e.g. [7].

**Figure 6.** Dimensionless blade differentiator $\xi$, based on [7].

Velocity:

$$w_\infty = \sqrt{c_m^2 + w_{\infty}^2}$$

Blade curvature angle:
For the value of the blade curvature angle and palisade throughput, the correction of the angle of attack is determined due to the work of the profile in the palisade. You can use the chart (Fig. 7)

![Figure 7. Angle of attack chart, based on [7].](image)

Then the calculations are repeated taking into account the angle of attack correction $\Delta \delta$:

$$\beta_e = \beta_\infty + \Delta \delta$$

Calculated again: the length of the profile chord, palisade throughput, dimensionless blade differentiator for the corrected angle $\beta_e$, speed $w_\infty$, angle of blade curvature and correction of the angle of attack. Iterate until we achieve consistency in the results of subsequent iterations. The next step is to enter the lift coefficient ($c_z$) as a function of circulation, relative speed $w_\infty$ and profile chord in the calculations

$$(c_z) = \frac{2\Gamma \cdot 1.6}{w_\infty l} \frac{1}{\xi}$$

Lift coefficient can be determined from:

$$c_z = \frac{\partial c_z}{\partial \delta} (\delta_\lambda - \delta_0)$$

where $\delta_\lambda$ - angle of attack for finite span blade, $\delta_0$ - angle of attack for zero lift force
Angle of attack of finite span blade is calculated by increasing the assumed angle of attack of infinity span blade by induced angle $\delta_i$:

$$\delta_\lambda = \delta + \delta_i$$  \hspace{1cm} (40)

$$\delta_i = 57.3 \frac{(c_z)}{\pi \lambda}$$ \hspace{1cm} (41)

The coefficient $\lambda$ for NACA profile is equal to 6.

At this stage, a certain value of angle $\delta$ should be assumed. The final value of this variable will be determined by iterative calculations.

After assuming the value of $\delta$ the induced angle value $\delta_i$ is calculated from formula (41) and the value of $\delta_\lambda$ angle from equation (40).

Next step is to determine the relative camber for the profile:

$$\frac{m}{l} = \frac{\sin^2 \left( \frac{\beta_s}{2} \right)}{\sin \beta_s} - \frac{1}{20} \tan \beta_s \frac{\beta_s}{5}$$ \hspace{1cm} (42)

Now the one has to assume the relative distance $\frac{L^*}{l}$ and maximum thickness $t^*$ (it should be remembered that due to strength reasons, the thickness of the profile along the blade height decreases - the profile at the hub is characterized by the greatest thickness).

Next the value of $\frac{L^*}{l}$ is calculated and derivative $\frac{\partial(c_z)}{\partial \delta}$.

$$\frac{\partial(c_z)}{\partial \delta} = 0.079 - 0.037 \frac{t^*}{l}$$ \hspace{1cm} (43)

Angle of attack at zero lift force:

$$\delta_0 = -83.3 \frac{m}{l} \left( 0.74 + \frac{L^*}{l} \right)$$ \hspace{1cm} (44)

The lift coefficient ($c_z$) is now determined on the basis of formulas (39) and (39).

If the result does not match, we correct the assumed angle $\delta$. After making the correction, the angle of inclination of the chord $\beta_e$ can be finally determined

$$\beta_e = \beta_\infty + \Delta \delta + \delta$$ \hspace{1cm} (45)

At this stage, you have all the data you need to plot the profiles and generate the rotor blade geometry.

3. Algorithm of stator design

The following method describes design process of a stator with relatively simple geometry that has obtained good results both in CFD simulation and during model tests.

When starting the design process, it is necessary to make a few assumptions:

- The shape of the stator hub and casing. In this case both the hub and casing are cylindrical.
- The number and thickness of blades. The number of stator blades is usually greater than the number of rotor blades by 1-2. Constant thickness of the blades can also be assumed - it simplifies the production process, without a significant reduction in efficiency.
- The angle of inclination of the inlet and outlet edges of the blade to the axis. The angle of inclination of stator inlet edge $\gamma_1$ should be chosen so that the rotor outlet edge and stator inlet edge are parallel. The angle of inclination of the outlet edge was set to $\gamma_2 = 90^\circ$.
• The distance between the rotor and the stator $X$ should be large enough to equalize the flow field behind the rotor, but at the same time not excessively high, as this increases the size of the pump. It is recommended that the distance $X$ is $0.1 - 0.15 \ l_s$, where $l_s$ is the length of the chord of the profile on the average streamline of the rotor [6].

• The stator length $L$ should be chosen in such a way as to ensure a smooth course of angle changes from $\alpha_4$ to $\alpha_5$, at the same time it must not be too long so as not to unnecessarily increase the size and weight of the pump. Finally, it is worth adding a straight section for additional straightening of the flow.

After choosing the shape of the stator hub and casing, number of blades $z_k$, thickness of blades $s_k$, length of the stator $L$ and angles $\gamma_1$ and $\gamma_2$, the next step is to calculate the meridional component of velocity at the rotor outlet:

$$c_{m2} = \frac{Q}{\pi (r_b^2 - r_A^2)}$$

Calculations are carried out separately for each streamline. In the case of a stator, three streamlines are sufficient (hub, center, casing). The slip factor is determined:

$$\varphi_2 = 1 - \frac{s_2Z}{2\pi r_s tan \beta_{s2}}$$

where $s_2$ is rotor blade thickness at the outlet. Meridional velocity:

$$c_{m3} = \frac{c_{m2}}{\varphi_2}$$

It is assumed that $c_{m3}$ is constant for each streamline. Next circumferential component is calculated:

$$c_{u3} = \frac{gH}{\mu \eta h}$$

Next it is assumed that $c_{m4} = c_{m3}$ and $c_{u4} = c_{u3}$. Therefore:

$$tg \alpha_4 = \frac{c_{m4}}{c_{u4}}$$

And the angle $\alpha_4$ is determined. In order to correct the angle $\alpha_4$, a narrowing coefficient $\mu_r$ and slip factor $\varphi_4$ should be assumed. After correction:

$$tg \alpha_4^* = \frac{tg \alpha_4 \mu_r}{\varphi_4}$$
And the determined angle is $\alpha^*$. The assumed slip factor should be checked

$$\varphi_4 = 1 - \frac{s_k z_k}{2 \pi r s \sin \alpha_4}$$

(52)

If necessary, correct the assumed value. The main problem in calculations is the assumption of the appropriate value of the $\mu_c$ coefficient. In the literature [7] one can find recommendations to take the coefficient value at the level of 1 - 1.5. However, the authors draw different conclusions. The coefficient should be set differently for each streamline, the lowest for the hub line. Its values should be in the range of 0.7 - 1.05.

4. An example of rotor and stator 3D geometry generation

This chapter will present the process of creating 3D geometry of a propeller pump with a significant head required (H over 13 m) and specific speed of approx. 140. It is a construction between a propeller pump and a diagonal pump, with quite unusual parameters. In addition, it should be possible to adjust the operating parameters by rotating the rotor blades around its axis.

Calculations were carried out in accordance with the algorithm presented in chapter 2 and 3.

Then coordinates of the profile outline for individual streamlines were generated. The relative camber $y$ coordinate is calculated on the basis of formulas (19) and (20). After multiplying by the chord value, we get the absolute coordinates: $X$ and $Ys$. Then, calculate for each $x$ coordinate: the derivative $dys/dx$, the angle $\Theta$ according to the formula (26), the coordinate $y$, based on the relationship (21), as well as the relative coordinates of the suction and pressure side edges of the profile from formulas (22-25). We get the absolute coordinates by multiplying the relative coordinates by the chord value. The profile nose can be rounded with a radius according to formula (27). It should also be remembered that the outline obtained from the formulas will have a sharp end, which, after generating the blade in 3D, must be rounded.

The set of coordinates should be saved and imported into the appropriate 3D geometry modeling program. Then incline the profile obtained at the calculated angle $\beta_e$ to the circumferential direction.

3D blade geometry is generated by creating cylindrical surfaces with diameters corresponding to the diameters of subsequent streamlines. Plotted profiles are wrapped on a suitable cylindrical surface (Fig. 9). Next, 3D sketches are created in which the wrapped profile profiles are converted to curves. Then create a surface by extruding the profiles (Fig. 10).

Figure 9. Profiles wrapped on cylindrical surfaces.

Figure 10. Blade surface generated by extruding the profiles.
The next step is to convert the resulting curved surface to a solid. Fig. 11 shows the rotor of the analyzed pump.

![Figure 11. The rotor of propeller pump.](image1)

Generating the 3D geometry of the stator was done as follows: three cylindrical surfaces were created according to the streamlines. The camber surface was an arc with an inlet angle $\alpha_4$ and an outlet angle $\alpha_5$. The blade was characterized by a constant thickness. A straight section has been added behind the arch. The blade profile formed was wrapped on a cylindrical surface, as in the case of a rotor (Fig. 12). Then, wrapped sketches were converted to curves in 3D sketches and extruded profiles were used to generate the blade. As in the case of the rotor, the upper and lower surfaces of the blades were added and all surfaces were combined into a solid (Fig. 13).

![Figure 12. The profiles wrapped on cylindrical surfaces – stator.](image2)  
![Figure 13. Blade surface generated by extruding the profiles – stator.](image3)

The overview of the stator is shown in Fig. 14.
The designed pump achieved good efficiency at the level of 86%, however, in the course of CFD analyzes some modifications were introduced, thanks to which the final efficiency was 90.1%. Only the flow system itself was taken into account, i.e. the rotor and the stator.

5. Numerical simulation conducted in ANSYS CFX
In order to plot the flow characteristic $H (Q)$ and power characteristics $P (Q)$ and efficiency $\eta (Q)$, the designed pump flow system was analyzed in the ANSYS CFX program. A numerical mesh of fluid flowing through the pump was generated, containing approximately 13.4 million hexahedra elements \[9\]. The division of the rotor and pump into sectors was abandoned, so that any phenomena occurring asymmetrically about the axis of rotation could be observed. A steady state flow has been assumed.

| Table 1. Numerical simulation settings. |
|----------------------------------------|
| Simulation type                        | Steady state                     |
| Inlet boundary condition               | Total pressure                   |
| Turbulence int4ensity at the inlet     | 5%                               |
| Outlet boundary condition              | Mass flow                        |
| Reference pressure                     | 1 atm                            |
| Turbulence model                       | k-ω SST                          |
| Fluid temperature                      | 25°C                             |
| Heat transfer                          | None (isothermal flow)           |
| Timestep                               | $1/\omega=8.2 \times 10^{-3}$ s  |

An analysis of the pump flow system was also carried out at three rotor blade angles other than nominal, i.e. $-2^\circ$ and $-4^\circ$. 

Figure 14. Overview of the stator.
6. Comparison between CFD and laboratory tests results
Then the pump model was reduced in the laboratory by factor 1: 2.25. A model pump was tested at various rotor blade angles. Then the measurement results for the model pump were converted into the characteristics of the analyzed pump:

\[
Q = Q_M \cdot \frac{n}{n_M} \cdot \left(\frac{D}{D_M}\right)^3
\]  
(53)

\[
H = H_M \cdot \left(\frac{n}{n_M}\right)^2 \cdot \left(\frac{D}{D_M}\right)^2
\]  
(54)

\[
P = P_M \cdot \left(\frac{n}{n_M}\right)^3 \cdot \left(\frac{D}{D_M}\right)^5
\]  
(55)

Efficiency was calculated by means of Byron Jackson formula:

\[
\eta = 1 - (1 - \eta_M) \cdot \left(\frac{D_M}{D}\right)^{0.165}
\]  
(56)

In Fig. 16-18 dimensionless characteristics of the pump obtained by both CFD and laboratory tests are presented.
Figure 16. $H(Q)$ chart – comparison between CFD and experiment.

Figure 17. $P(Q)$ chart – comparison between CFD and experiment.

Figure 18. $\eta(Q)$ chart – comparison between CFD and experiment.
The H (Q) characteristics for the 0 (nominal), -2 ° and -4 ° angles obtained in the numerical simulation do not differ much from the characteristics obtained as a result of the measurements. For the nominal angle, the values of the head begin to differ significantly at capacities less than 70% of the rated capacity. However, working so far from the nominal point is not recommended due to the increasingly intense flow disturbances [10].

It should be noted that the pressure measurement was located behind the elbow draining the liquid from the stator. For this reason, the efficiency determined based on the measurement is burdened with energy loss in the elbow. The outlet elbow could not be modified; therefore its optimization was not undertaken. However, to be able to compare the results, the discharge element was mapped in the simulation and the efficiency was determined once more. An efficiency of 85% was obtained, compared to the efficiency determined for the flow system alone of over 90%, this is a much worse result. Therefore, it is extremely important to pay attention to the installation conditions of the pump, the shape and optimization of the elements draining and supplying liquid to the flow system [11-13].

For the nominal angle, the efficiency values obtained from measurements and simulations are in good agreement, for measurements they are even slightly higher. For the -2 ° angle, numerical analysis showed slightly overestimated efficiency, while for the -4 ° angle it was underestimated. It can be stated that the prediction of real efficiency based on numerical simulations is justified, although it should be remembered that the calculation of efficiency from model pump tests is also subject to some error [14].

7. Conclusion

Based on the numerical analysis of the water flow through the propeller pump and the analysis of the measurement results obtained at the test station, the following conclusions can be drawn regarding the design methods of rotor and stator of propeller pump:

- The head obtained from the first versions of the construction was about 25% lower than expected. The introduction of the power shortage factor, i.e. increasing the circulation around the blades by 25%, allowed achieving the planned pump head.
- Stator design was crucial to obtain good efficiency result.
- The inlet angle $\alpha_4$ should be different for each streamline. For streamline at the hub, this angle was reduced several times to prevent vortices in the stator. The narrowing coefficient $\mu_r$ can take values from a range wider than that reported in the literature, from 0.70 to 1.1.
- In case of the stator, better performance was achieved with a cylindrical hub and casings than with conical surfaces that widen the cross-section of the channel. It was important to keep the speed at an appropriate level to prevent vortices appearance in the stator.
- Pump installation conditions are also very important. The effort to optimize the impeller and steering wheel shape can be lost by significant losses of pressure in fluid supply and drainage components.

The presented algorithm allows generating geometry that achieves good efficiency, but on the other hand carrying out numerical simulations allows localizing and eliminating weaker points - thanks to this it is possible to achieve efficiency of over 90% even for constructions with unusual parameters.

References

[1] Jędral W 2001 Pompy Wirowe (Warszawa: PWN)
[2] Varchola M, Hlbocan P 2016 Hydraulicky navrh axialneho stroja (Bratislava: Slovenska technicka universita v Bratislave)
[3] Ganeshan V K 2018 Analysis and Optimization of Axial Flow Turbo Pump International Research Journal of Engineering and Technology (IRJET) 05(01) 1397
[4] Łazarkiewicz Sz T A 1968 Pompy Wirowe ( Warszawa: WNT)
[5] Rui Z R 2013 Numerical analysis of cavitation within slanted axial-flow pump Journal of Hydrodynamics 25(5) 663
[6] Gulich J F 2010 Centrifugal Pumps (Berlin: Second Edition, Springer – Verlag)
[7] Stepanoff A 1959 Radial und Axialpumpen (Berlin: Springer – Verlag)
[8] Nestmann P S 2011 Axial flow impeller WORLD PUMPS 36
[9] Fu-Jun L Y-J 2007 Numerical investigation of performance of an axial flow. Journal of Hydrodynamics 19(6) 705
[10] Nikola M V 2012 Possible efficiency improvement by application of various operating regimes for the cooling water pump Thermal Science 16(1) 263
[11] Street M L, King Jr 2006 Pump seal problems in cooling water system Peerless Pump Company TECHNICAL INFORMATION 1-6
[12] Zahorulko A 2015 Experimental investigation of mechanical properties of stuffing box packings Sealing Technology 8 30244 7-13
[13] Vigneshkumar R P Y 2018 Reliability improvement of cooling water pump International Journal of Mechanical and Production 8(3) 755
[14] Knopkar A R, Mavros P, Ranade V V, Bertrand J 2004 Simulation of flow generated by axial flow impeller Chemical Engineering Research and Design 82(A6) 737