Determination and generalization of the effects of design parameters on Francis turbine runner performance

Ece Ayli\textsuperscript{a}, Kutay Celebioglu\textsuperscript{b} and Selin Aradag\textsuperscript{a}

\textsuperscript{a}Department of Mechanical Engineering, TOBB University of Economics and Technology, Ankara, Turkey; \textsuperscript{b}TOBB ETU Hydro Energy Research Laboratory, TOBB University of Economics and Technology, Ankara, Turkey

\textbf{ABSTRACT}

The runner design is the most challenging part of the turbine design process. Several parameters determine the performance and cavitation characteristics of the runner: the metal angle (flow beta angle), the alpha angle, the blade beta angle, the runner inlet and outlet diameters, and the blade height. All of these geometrical parameters need to be optimized to ensure that the head, flow rate and power requirements of the system are met. A hydraulic designer has to allocate time to optimize these parameters and should be experienced in carrying out the iterative design process. In this article, the turbine runner parameters that affect the performance and cavitation characteristics of designed turbines are examined in detail. Furthermore, turbines are custom designed according to the properties of hydroelectric power plants; this makes the design process even more challenging, as the rotational speed, runner geometry, system head and flow rate vary for each turbine. The effects of the design parameters are examined for four different turbine runners specifically designed and used in actual power plants in order to obtain general results and generalizations applicable to turbine design aided by computational fluid dynamics (CFD). The flow behavior, flow angles, head losses, pressure distribution, and cavitation characteristics are computed, analyzed, and compared. To assist hydraulic designers, the general influences of these parameters on the performance of turbines are summarized and empirical formulations are derived for runner performance characterization.

\section{1. Introduction}

The most important component of Francis-type hydraulic turbines is the runner, which controls the power generation and cavitation characteristics. Computational fluid dynamics (CFD) has become an effective tool for analyzing fluid flow over the past decades, especially in turbomachinery (Ayli et al., 2015; Round, 2004). Runner design can be considerably improved using experimental and CFD-based techniques (Daneshkah & Zangeneh, 2010).

There are several case studies in the literature relating to hydraulic turbine runner design based on CFD (Anup, Thapa, & Lee, 2014; Gohil & Saini, 2014; Patel, Desai, Chauhan, & Charnia, 2011); however, there are no generalizations on the effects of all the related parameters on the design performance of runners. There are several runner design parameters which according to Ayancik, Celebioglu, and Aradag (2014) affect the blade shape and turbine performance, and researchers are interested in understanding the general effects of design parameters that apply to all Francis turbine runners.

Daneshkah and Zangeneh (2010) studied the effects of stacking on cavitation and blade loading; the low-pressure region on the suction side of the blade is eliminated in their design and a cavitation free runner is obtained with an optimum stacking angle. Farell, Arroyave, Cruz, and Gulliver (1993) investigated the effect of the rotational speed on the efficiency, power and flow rate of turbines. Flow discharge was found to be a decreasing function of rotational speed for low specific speeds (suitable for Francis turbine applications) and for higher specific speeds (suitable for axial turbine applications), while the flow rate was found to increase with increasing rotational speed. The hydromechanics of variable speed turbines are explained briefly in Ardanuy, Wilhelmi, Mora, and Perez (2006).

The process of designing high-speed turbines using CFD is presented in Obrovsky, Krausowa, Spidla, and Zouhar (2013), wherein they use the auto-optimization method for runner design in order to observe the effects of different runner parameters on the performance and cavitation characteristics of hydraulic turbines.
Kurosawa, Lim, and Enomoto (2010) performed virtual model tests of a Francis turbine. A high-accuracy prediction method was developed for Francis-type turbines and the entire flow was computed to reduce numerical errors. Francis turbines with three different specific speeds were simulated and tested, and model tests were performed on a test rig at a hydraulic research laboratory. The comparison of their results verified that the critical cavitation coefficient can be predicted with high accuracy.

Zhang and Zhang (2012) performed a numerical analysis of cavitating turbulent flow in a high-head Francis turbine at partial load operation. The computational grid, which consists of 8 million elements, was generated using ICEM CFD for the spiral case, stay vanes, guide vanes, runner and draft tube. The analysis was performed using a $k-\omega$ shear stress transport (SST) turbulence model in OpenFOAM. According to the authors, it is important to observe the volume fraction of water vapor in the cavitating flow. The tendencies of the cavitating flow in the runner and the draft tube agree well with the empirical results for the turbine.

Qian, Yang, and Huai (2007) performed 3D unsteady multiphase flow simulations for the whole Francis turbine. They focused on predicting pressure fluctuations with respect to the spiral case, the guide vane and the runner. The pressure fluctuations were analyzed using a fast Fourier transform (FFT) and the amplitude spectrum was generated. The experimental results corroborate the computational ones and the relation between pressure fluctuations and air admission was analyzed. It was found that the dominant frequency is determined by the runner frequency, the geometry of the runner and the draft tube.

Su, Li, Li, Wei, and Zhao (2012) solved the three-dimensional turbulent flow in Francis turbines computationally using a large eddy simulation (LES) method. They used an unstructured mesh for the spiral case and the runner, while structured meshes were utilized for the remaining components. LES models reproduce the efficiency better than RANS models, according to their findings. They also found that cavitation takes place at the suction side of the blade under partial load conditions.

Every hydroelectric power plant (HEPP) requires a custom design for a turbine in order to operate at maximum efficiency based on its head and flow rate. The rotational speed of the turbine, runner geometry, head and flow rate affect the design. In this article, the effects of the design parameters are investigated using the results of four different turbines specifically designed for four different hydroelectric power plants in order to make generalizations and obtain correlations applicable to turbine design and examine the general characteristics. Flow behavior, flow angles, head losses, pressure distribution and cavitation characteristics are computed, analyzed and compared for the designed turbine runners. The general influences of these parameters on the turbine performance are summarized and empirical formulations are derived for runner performance characterization. Since every power plant needs a unique turbine design, the challenge is to obtain generalizations and generalized formulations based on several cases that are applicable to a whole range of turbines.

The runner design process is affected by several parameters because of the complexity of its geometry, as well as three-dimensional rotational nature of the flow. The most important parameters are the rotational speed of the turbine $n$, the guide vane height $b_0$, the runner inlet and outlet diameters $D_1$ and $D_2$, the alpha angle $\alpha$, the blade beta angle $\beta$, the flow angle beta $\beta_f$, and the theta angle $\theta$ (Figure 1). These parameters are modified to ensure that the power, efficiency and hydraulic performance of the system are maximized. A parametric study was performed to determine the effects of the parameters on the performance of the turbine.

**Figure 1.** Radial-axial flow turbine runner (a) velocity parallelograms and (b) meridional flow.

Note: In this figure 1, $V_{1u}$ is tangential velocity component, $V_{1m}$ is meridional velocity component, $V_1$ and $V_2$ are inlet and outlet absolute velocity components, respectively. $u_1$ and $u_2$ are peripheral inlet and outlet velocity components, respectively. $\omega_1$ and $\omega_2$ are relative speed at the inlet and outlet. $D_{1d}$ denotes runner inlet diameter and $D_{2d}$ denotes runner outlet diameter.
Table 1. Parameters of the parametric study.

| Parameter                          | Optimization range                      |
|------------------------------------|----------------------------------------|
| Runner outlet diameter, $D_1$      | 0.6–1.0 m                               |
| Inlet beta angle, $\beta$          | $-4^\circ + \beta_{\text{nom}} < \beta < 20^\circ + \beta_{\text{nom}}$ |
| Outlet beta angle, $\beta$         | $-4^\circ + \beta_{\text{nom}} < \beta < 8^\circ + \beta_{\text{nom}}$ |
| Lean angle, $\delta$               | $-10^\circ + \delta_{\text{nom}} < \delta < 10^\circ + \delta_{\text{nom}}$ |
| Rotational speed, $\omega$         | 750–1250 rpm                            |

design parameters on performance. Table 1 shows the parameters and their ranges.

2. Kinematics of Francis turbines

The torque on the shaft is equal to the change in angular momentum of the water flow. Equation (1) below is obtained by using the angular momentum equation for the inlet to the outlet of the runner. The sum of the moments of external forces about the axis of rotation is as follows:

$$\rho Q (V_2 u_2 - V_1 u_1) = \sum M_0.$$  \hspace{1cm} (1)

Figure 2 presents the velocities at the leading and trailing edges of the runner blades. Torque is only generated when runner blades generate circulation in the runner. The generated power can be calculated using the moment equation and angular velocity of the runner:

$$N = M \omega = \rho g H Q \eta,$$  \hspace{1cm} (2)

where $H$ is the head of the turbine and $\eta$ is the hydraulic efficiency. The main energy equation of turbines (Euler equation) is obtained substituting for $M$ in Equation (2):

$$H \eta = \frac{1}{g} (u_1 v_1 \cos \alpha_1 - u_2 v_2 \cos \alpha_2).$$  \hspace{1cm} (3)

Equation (3) can be written as follows using the expressions for circulation:

$$H \eta = \frac{\omega}{2 \pi g} (\Gamma_1 - \Gamma_2).$$  \hspace{1cm} (4)

Hydraulic energy is related to the mechanical energy which is absorbed by the runner as shown in Equation (3). The angles at which water enters and exits the runner are also important parameters that affect hydraulic performance. The position of the guide vane determines the angle of the water inlet of the runner; therefore, the guide vane angle is a major parameter as well. Changing the guide vane angle changes the efficiency.

The flow rate $Q$, rotational speed $n$ and head $H$ are the preset operating conditions of the turbine. It is possible to determine the components of the velocity parallelograms using these known parameters and the dimensions of the runner (Figure 1(a)). In Figure 1(a), $D_{1d}$ is the inlet of the blade edge and $D_{2d}$ is the outlet of the blade edge.

The transportation velocity is expressed as:

$$u = \frac{\pi D n}{60},$$  \hspace{1cm} (5)

where $n$ is the rotational speed and $D$ is the inlet or outlet of the blade edge. The meridional component $V_m$ can be calculated using continuity equation and is defined as:

$$v_m = \frac{Q}{\pi D_{1d} \omega b_0},$$  \hspace{1cm} (6)

where $b_0$ is the height of the inlet opening of the runner. The peripheral component $V_{1u}$ is calculated from constant circulation condition and is equal to:

$$v_{1u} = v_0 \frac{D_{02}}{D_{1d}}.$$  \hspace{1cm} (7)

Q: Flow rate
r: Fluid density
$\alpha$: Incidence angle
$V_u$: Tangential fluid velocity
$U$: Transportation velocity
$\omega$: Relative velocity
T: Torque
P: Power output
Finally, $V_1$ and $V_2$ can be found by calculating the meridional and peripheral components as follows:

$$v_i = \sqrt{v_{im}^2 + v_{ip}^2}. \quad (8)$$

### 3. Properties of the designed turbines and design methodology

The design methodology used for the CFD-aided design of hydraulic turbines is explained in detail in Ayli et al. (2015). In the present study, parameter studies were performed for four different existing HEPP turbine runners (Akin et al., 2013; Ayancık et al., 2014; Ayli et al., 2015) in order to obtain generalized results that are valid for all Francis turbine runners. The turbines investigated in this study are numbered Turbine 1 to Turbine 4 for ease of reference and their general performance values are presented in Table 2.

Numerical simulation methods and parameters such as the numerical scheme, computational grid quality, boundary conditions, and proper turbulence model are important factors that affect the accuracy of the predicted hydraulic performance. A converged and mesh-independent solution was obtained for the CFD-aided design of each turbine. Six different meshes were prepared for a mesh-independency study. The grid-independency test results are presented in Figure 3 with the help of the runner efficiencies and power values. For all turbine designs, 1 million elements are used for one blade and 15 million mesh elements are used for the whole runner simulation. Approximately 22 million

| Turbine HEPP | Turbine 1 Atakoy | Turbine 2 Koprubasi | Turbine 3 Yuvacik | Turbine 4 Buski |
|--------------|------------------|---------------------|------------------|----------------|
| Rotational speed, $\omega$ (rpm) | 78.54 | 104.72 | 104.72 | 104.72 |
| Flow Rate, Q (m$^3$/s) | 4.25 | 3.75 | 2.50 | 2.00 |
| Head, H (m) | 66.00 | 126.00 | 45.00 | 78.00 |
| Efficiency, $\eta$ at BEP | 91.40 | 92.00 | 92.00 | 93.00 |

Note: BEP, Best efficiency point.

Figure 3. Mesh independency study for (a) Turbine 1, (b) Turbine 2, (c) Turbine 3, and (d) Turbine 4.
elements are used for full turbine analyses, utilizing a computational cluster.

In the full turbine analyses, the stay vanes, guide vanes and runner are meshed with an H/J/C/L grid using the ANSYS TurboGrid module, and the spiral case and draft tube are meshed using hexahedral mesh elements. The mass flow inlet and pressure outlet are used as boundary conditions, and the pressure inlet and mass flow outlet boundary conditions are utilized for the analysis of the runner.

Turbulence models are used while solving the governing equations computationally. The governing equations for the viscous flow of a Newtonian fluid are given below. Incompressible Navier–Stokes equations are modeled using the $k$-$\varepsilon$ turbulence model:

$$
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0, \quad (9)
$$

$$
\frac{D(u_i)}{Dt} = \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j^2} + F_{i}, \quad (10)
$$

$$
\frac{\partial}{\partial t} \left[ \rho \left( e + \frac{1}{2} u_i u_i \right) \right] + \frac{\partial}{\partial x_j} \left[ \rho u_j \left( h + \frac{1}{2} u_i u_i \right) \right] = - \frac{\partial}{\partial x_j} (u_i t_{ij}) + \frac{\partial q_j}{\partial x_j}. \quad (11)
$$

The turbulence model test results for Turbine 4 are presented in Figure 4 with the help of the efficiency, power and head values. The results for the $k$-$\varepsilon$, $k$-$\omega$ SST and renormalization group $k$-$\varepsilon$ turbulence models are compared. A similar trend is obtained with all the turbulence models tested according to the quantitative and qualitative results (Figures 4 and 5). The $k$-$\varepsilon$ turbulence model was chosen, taking the time cost into account. As these types of problems are not dominated by boundary layers, the $k$-$\omega$ SST model presents a similar flow behavior to that of the $k$-$\varepsilon$ turbulence model (Table 3).

A multi-frame reference (MFR) model is used for the full system analysis, with five components (Keck & Sick, 2010). The runner is attached to the guide vane and draft tube by the frozen rotor interface. Mathematical models were developed to observe the rotor–stator interaction. The MFR model is a steady state approximation. In the frozen rotor approach, the motion of the moving part is frozen in a specific position and the instantaneous flow field with the rotor in the given position is observed (Keck & Sick, 2010).

4. Results

4.1. The effect of the lean angle on runner performance

The lean angle is defined as the blade’s lean, namely, the degree of hub or shroud line shifted from the original position (Gjøsæter, 2011). The lean angle is one of the most important parameters for controlling the pressure balance in the runner. Figure 5 shows the reference design, which has no lean (0° lean angle), and the new designs, modified from reference design, are shown. A 5° and a 10° lean are given both from the hub and the shroud, in both the direction of rotation and the reverse direction of rotation. All other runner design parameters are kept constant to test the effect of the lean angle on runner performance.

4.1.1. Leans from the hub layer

As the linear lean is defined on the blade, when the shroud layer is kept constant the blade is twisted from the hub in the direction of rotation and the reverse direction of rotation. As shown in Figure 6, the efficiency and power increase as the lean decreases in the direction of rotation, and they keep increasing up to a 5° lean in the reverse direction of rotation.

The location at which the water hits the leading edge of the blade changes because of the lean angle. In Figure 7, vector plots are shown for different lean angles. When the blade leans in the reverse direction of rotation, the water hits the suction side of the blade; when the blade leans in the direction of rotation, the water hits the pressure

![Figure 4. Turbulence model comparison for Turbine 4 using (a) the $k$-$\omega$ SST turbulence model, (b) the $k$-$\varepsilon$ turbulence model, and (c) the RNG $k$-$\varepsilon$ turbulence model.](image-url)
side of the turbine runner blade. The results show that when the former load situation results in an increase in efficiency and power.

Figure 8 shows the pressure loads for the given cases. Although the efficiency and power values are higher in the cases where there is a lean in the reverse direction of rotation, the possibility of exposing the blade to leading-edge cavitation increases. When there is a lean in the direction of rotation, the pressure distributions on the suction side and pressure side are more uniform. Likewise, the blade loads and static pressure contours illustrate the negative pressure zones on the blade (Figure 9). In the original design, the negative pressure zone is observed on the suction side of the leading edge. This negative pressure zone expands and makes the blade prone to cavitation damage when there is a lean in the reverse direction of rotation. Contrarily, this zone is reduced when there is a lean in the direction of rotation.

4.1.2. Leans from the shroud layer
The lean angle versus power and efficiency for the case where a lean is introduced from the shroud layer is shown in Figure 10. The maximum performance is generated for a 10° lean angle, while a 5° lean from the hub and a −5° lean from the shroud result in the same blade profile.
However, although they have the same blade profile, the lean from the hub and the lean from the shroud result in a different performance.

As with the lean from the hub, when the blade is twisted in the reverse direction of rotation the water hits the suction side, and when there is a lean in the direction of rotation the water hits the pressure side. The blade loading profiles for different lean angles are shown in Figure 11. Leans from the shroud have much more influence on the blade loading profiles compared to leans from the hub. Leans in the reverse direction of rotation make the blade prone to cavitation damage, especially for a lean angle of 5°. Increasing the lean in the direction of rotation improves the cavitation characteristics.

In Figure 12, static pressure contours are shown for cases with leans from the shroud. It is seen that independent of the direction of rotation, the negative pressure zone on the shroud suction side reduces. Even though, the smallest pressure value does not change considerably, the shape of the low-pressure zone changes and the zone shrinks.

Based on the results of the effect of the lean angle on runner performance, flow behavior and pressure distribution, the following conclusions are drawn:

- If there is a negative pressure zone which causes a reduction in the generated power and efficiency, introducing a lean from the hub in the direction of rotation can result in an increase in performance.
- When there is a negative pressure gradient in the shroud layer, defining a rotation from the shroud layer independent of rotation direction is a sufficient solution.
- The incidence angle can be settled by defining the lean angle.
- Even though leans from the hub layer and leans from the shroud layer can provide the same geometrical blade structure, their performance is different.
Figure 9. Static pressure contours for lean angles from the hub of (a) 0°, (b) −5°, (c) −10°, (d) +5°, and (e) +10°.

Figure 10. Power and efficiency versus lean angles from the shroud.

4.2. The effect of the metal angle (leading-edge blade beta angle) on runner performance

The metal angle of the blade (beta angle) is one of the parameters that is important to defining the runner blade geometry. The direction of the runner blades at a given point is determined by the blade beta angle, which is the angle between the chamber of the blade and the tangent of the circle that is drawn from the center of the runner. In this part of the study, the effects of the blade beta angle on runner performance and cavitation characteristics are examined. Four runner profiles were studied in order to understand the effect of the beta angle on different turbine runners (Table 2).

As the blade beta angle decreases, the length of the blade increases and the blade chamber decreases; conversely, as the blade beta angle increases, the length of the blade decreases and the blade chamber increases (Figure 13).

Figure 14 shows the beta angle versus power and energy coefficient curves for the four turbine runners. The energy coefficient \( \psi \) is computed using the following relation:

\[
\psi = \frac{H}{(u_1^2/2g)},
\]

where \( u_1 \) is the peripheral velocity of the runner, which is equal to \( \pi D_1 n \), and \( n \) is measured in units of 1/s. It is observed that the leading-edge beta angle has a direct effect on the power and the head. The change in power versus the beta angle shows a parabolic distribution for all four turbines. The generated power increases up to a critical leading-edge beta value, after which it starts to drop. When the original beta angle distribution of the turbine runners and the critical beta values of the runners are investigated in detail, together, it is verified that for all turbines the critical beta value is different from the others. This implies that the critical leading-edge beta
angle which controls the generated power and head is unique for all of the turbines. For low beta angles, friction losses increase because of the length of the blade, as can be seen from the head distributions. This explains why the energy coefficient values decrease for low beta angles. The leading-edge beta angle does not have a remarkable
influence on the hydraulic efficiency, except in the case of Turbine 1 (Figure 15).

Figure 16 shows the peripheral inlet velocity versus the change in the beta angle. The peripheral velocity distribution reveals that the inlet circulation generated at the guide vane trailing edge rises up to a critical beta value which increases the generated power. A similar trend is observed in the generated power; therefore, it is understood that the leading-edge beta has a direct effect on the peripheral inlet velocity which controls the generated inlet circulation.

Thoma number curves are obtained using pressure distributions to understand the effect of the leading-edge beta on cavitation. A Thoma number graph is shown for Turbine 1 in Figure 17. The Thoma number $\sigma$ is computed using the following relation:

$$\sigma_{plant} = \frac{(P_{amb} - P_{va})/\rho g - Z_s}{H},$$

(13)
\[ \sigma_p - \sigma_i = \left( \frac{P_i - P_v}{\rho g} \right) \left( \frac{1}{H_n} \right), \]  

where \( P_i \) is the pressure at the center of element \( i \), \( P_v \) is the vapor pressure determined by site conditions, \( H_n \) is the net head, \( \sigma_p \) is the plant sigma and \( \sigma_i \) is the sigma calculated at the center of element \( i \). Cavitation occurs when the pressure is less than the vapor pressure.

The leading-edge beta angle has no significant effect on the cavitation characteristics for all of the turbines. The leading-edge beta causes a change in blade geometry. A vortex region arises due to the rise of the blade chamber for high leading-edge beta angles, while the flow is uniform and follows the blade surface for low leading-edge beta angles (Figure 18).

The effects of the leading-edge blade beta angle on runner performance are as follows:

- The leading-edge beta angle has an impact on the generated power rather than the cavitation characteristic of the turbine. Each of the turbines has a unique critical leading-edge beta angle. The generated power increases up to this critical value, after which it starts to decrease.
Figure 17. Thoma number distribution for Turbine 1.

Figure 18. Velocity vector plot at the hub of runner blade (a) blade with low leading edge angles (b) blade with high leading edge angles.

Figure 19. Velocity components versus $D_2/D_1$ for (a) the peripheral velocity and (b) the meridional velocity.
Changes in the leading-edge beta angle also alter the blade geometry. Friction losses increase for higher beta angles. On the other hand, the flow becomes more uniform as the blade chamber decreases. Therefore, there is a balance between the blade length and the blade chamber which balances the head losses and flow structure for the critical beta angle.

The leading-edge beta value can be adjusted to control the generated power with a negligible change in efficiency and cavitation characteristics after reaching the required cavitation characteristics and efficiency values.

4.3. The effect of the runner outlet diameter on runner performance

Changing the runner outlet diameter has a direct impact on velocity components and, consequently, on velocity triangles, as can be seen from Equations (1) to (8). Increasing the runner outlet diameter increases the peripheral velocity component $u_2$ and decreases the meridional velocity component $v_{2m}$ due to the continuity equation (Figure 19). The absolute outlet velocity $v_2$ changes because of the change of meridional and peripheral velocity components (Figure 20(a)). The absolute outlet velocity decreases as the outlet diameter increases for all four turbines.

A decrease in the outlet meridional velocity component $V_{m1}$ causes a change in the inlet meridional velocity component. Thus, a change in the outlet diameter causes a change in the relative velocity at the inlet (Figure 20(b)). The effect of the diameter on the efficiency is shown in Figure 21. The efficiency versus diameter distribution exhibits the same tendencies for all four turbines. The maximum efficiency and power are obtained in the range of $0.8 < D_2/D_1 < 0.9$.

Figure 20. Velocity components versus $D_2/D_1$ for (a) the absolute outlet velocity component and (b) the absolute inlet velocity component.

Figure 21. Efficiency versus $D_2/D_1$ curves for all of the four turbines.
It can be claimed based on the efficiency and power distribution for all turbines that if an outlet-to-inlet diameter ratio is chosen in the range of $0.8 < D_2/D_1 < 0.9$ then the design process will start with maximum efficiency and power. When increasing the diameter, according to the continuity equation as the minimum pressure values increase the cavitation possibility decreases due to a rise in local velocities. Decreasing the diameter also increases the tendency for cavitation to occur. In Figure 22, the Thoma number contours are given for Turbine 1. For Turbine 1, the sigma plant is 0.13951. When the local sigma value is higher than the sigma plant, cavitation will occur on the runner blades.

4.4. The effect of the rotational speed on Francis turbine kinematics

A total of 160 full turbine analyses were performed to investigate the effect of the rotational speed on power, head and efficiency (Figure 23). This study was only performed for Turbine 1.

The flow beta and alpha angles were investigated to understand the velocity triangle behavior with changes in rotational speed, where alpha $\alpha$ is the angle between the tangential and absolute velocity vectors and flow beta
β is the angle between the transportation and relative velocity, which is called the ‘angle of attack’. In Figure 24, the velocity triangles at the leading edge and the trailing edge of the runner blade are shown schematically for different rotational speeds.

With an increase in rotational speed, \( w_1 \) deflects to the opposite direction of rotation and the angle of attack becomes smaller than the metal angle. For high rotational speed, \( \beta_{\text{flow}} < \beta_{\text{blade}} \). It can be seen in Figure 25 that the maximum power and efficiency are obtained between 800 rpm and 1000 rpm for \( \beta \approx \delta \) and a shock free flow occurs. In Figure 26, velocity vectors are shown for different rotational speed values. When \( \beta_{\text{flow}} > \beta_{\text{blade}} \) (500 rpm and 800 rpm), a vorticity region occurs at the suction side of the runner blade inlet edge. When \( \beta_{\text{flow}} < \beta_{\text{blade}} \), a vorticity region occurs at the pressure side of the runner blade inlet edge. Vorticity regions cause energy losses which reduce the generated power and the turbine efficiency.

In Figure 27(b), velocity triangles at the trailing edge of the runner blade are shown schematically for different rotational speeds. When the rotational speed increases, \( \alpha \) angle also decreases as \( n \) increases. The best operating conditions are achieved when \( \Gamma_2 \approx 0 \), when efficiency is taken into account. For this purpose, the outlet alpha angle should be nearly 90° in theory, and the circulation and efficiency versus rotational speed are shown in Figure 27. It is seen in this figure that the maximum efficiency and power are not obtained under the condition of \( \Gamma_2 \approx 0 \). When a circulation of 3 m²/s
is released at the outlet, which is not prone to cavitation damage as it has a small frequency, a little swirl starts in the draft tube that prevents boundary-layer detachments and separation. Therefore, the total hydraulic efficiency increases when little swirl is released into a draft tube, as the draft tube recovery factor increases and the losses decrease with the help of the generated swirl.

4.5. Development of a correlation for the flow rate and rotational speed

It is shown in Figure 28 that an increase in rotational speed results in a decrease in the flow rate. A theoretical equation is presented in this section for the relation between the flow rate and the rotational speed. Firstly, the absolute inlet and outlet velocities are written in terms of the flow rate. The velocity parallelogram equation is used for the outlet velocity for the absolute inlet velocity circulation equation:

\[ v_1 \cos \alpha_1 r_1 = v_0 \cos \alpha_0 r_0, \quad (15) \]
\[ Q = AV = v_0 \sin \alpha_0 (2\pi r_0) (b_0), \quad (16) \]
\[ v_1 \cos \alpha_1 = \frac{v_0 \cos \alpha_0 r_0}{r_1}, \quad (17) \]
\[ v_1 \cos \alpha_1 = \frac{Q}{2\pi r_1 b_0 \tan \alpha_0}, \quad (18) \]
\[ v_2 \cos \alpha_2 \pm w_2 \cos \beta_2 = u_2, \quad (19) \]
Figure 28. Variable rotational speed characteristics of a 78 m head turbine.

Figure 29. Rotational speed versus (a) head and (b) beta angle.

Figure 30. Flow rate versus rotational speed for CFD and empirical results.
Equations (18) and (21) are written into the Euler equation below, after which Equation (23) is obtained:

\[ gH \eta = u_1 v_1 \cos \alpha_1 - u_2 v_2 \cos \alpha_2, \]  
\[ Q = \frac{r_2 \omega_2 + (gH/\omega_2 r_2) - (1/2 \pi b_0 \tan \alpha_0 r_2) + (1/A_2 \tan \beta_2)}{(1/2 \pi b_0 \tan \alpha_0 r_2) + (1/A_2 \tan \beta_2)}, \]  

where the outlet beta angle ($\beta_2$), rotational speed ($\omega = u \times r$), head ($H$) and efficiency ($\eta$) are dependent on the flow rate. In this case, there is no direct relationship between the flow rate and the rotational speed since there is more than one dependent variable in the equation. The head variation's dependency on the rotational speed is shown in Figure 29(a), from which it can be observed that the head is nearly constant between 800 rpm to 1200 rpm. The efficiency change is negligible in this range of rotational speed values (Figure 29).

The beta angle versus the rotational speed curve was plotted and curve fitting was performed for a rotational speed range of 800 rpm to 1200 rpm. Equation (24) is obtained as:

\[ \beta_2(\eta) = (10^{-10} \times n^4) - (4 \times 10^{-7} \times n^3) + (0.0004 \times n^2) - (0.0484 \times n). \]  

Then the equation below is obtained for constant head and efficiency range by substituting Equation (15) into Equation (1):

\[ Q = \frac{r_2 \omega_2 + (gH/w_2 r_2)}{(1/2 \pi b_0 \tan \alpha_0 r_2) + (1/A_2 \tan \beta_2)} \times \frac{1/2 \pi b_0 \tan \alpha_0 r_2) + (1/A_2 \tan \beta_2)}{10^{-10} \times n^4 - (4 \times 10^{-7} \times n^3) + (0.0004 \times n^2) - (0.0484 \times n)}. \]  

The flow rate versus the rotational speed is obtained with the assumption of 100% efficiency and a 74 m head using Equation (25). Figure 30 shows that the efficiency has a negligible effect when the CFD and empirical results are compared. Even though $\beta_2(\eta)$ is specific to this turbine runner, the general behavior (the outlet beta increases with the rotational speed) is universal for all Francis-type turbines. Therefore, generalized results can be obtained using Figure 30. The flow that enters into the turbine runner per unit time decreases as the rotational speed increases.
The speed-discharge characteristic of Turbine 4 is depicted as a hill chart in Figure 31. Isoclines of hydraulic efficiency are also plotted on the same graph, from which it can be observed that the maximum efficiency is obtained at the design rotational speed.

5. Discussion and conclusions

In this paper, hydraulic design parameters are varied within a defined range and their effects on the performance and cavitation characteristics of a hydraulic turbine are evaluated. Four turbine runners have been examined in detail to obtain universal characteristics for the varied parameters. A summary of the findings of this study is as follows:

- If there is a negative pressure zone which causes a reduction in the generated power and efficiency, introducing a lean from the hub in the direction of rotation can result in an increase in performance.
- Even though leans from the hub layer and leans from the shroud layer can provide the same geometrical blade structure, the resulting generated power and efficiency are different.
- The leading-edge beta angle has an impact on the generated power rather than the cavitation characteristic of the turbine. Each of the turbines has a unique critical leading-edge beta angle. The generated power increases up to this critical value, after which it starts to decrease.
- The leading-edge beta value can be adjusted to control the generated power with a negligible change in efficiency and cavitation characteristics after reaching the required cavitation characteristics and efficiency values.
- If an outlet-to-inlet diameter ratio is chosen in the range of $0.8 < D_2/D_1 < 0.9$, the design process will start with maximum efficiency and power.
- There is an empirical relationship between the flow rate and the rotational speed, and according to the obtained correlation the flow rate decreases with higher rotational speeds.
- Although, according to theoretical equations, the maximum power is obtained when the outlet alpha angle is $90^\circ$, it is seen that, in practice, a little swirl in the runner outlet and draft tube inlet increase the performance and, as a result, the generated power.

CFD is a powerful tool for estimating the flow behavior in hydraulic turbines and this tool has been used in this study. However, the next step is to test the designs using experimental methods. Therefore, a test center, which is 20 m high and with a base area of 600 m$^2$, has been constructed and is currently being used at the authors’ test center. Although the designs presented here are installed in actual power plants without tests, the test center will be used for design improvements and certification for future designs as a complementary tool for CFD. In terms of computational efforts, time-dependent simulations and the detection of the time-dependent behavior of cavitation are the steps that will next be undertaken in this ongoing work.

Funding

This work is supported by the Infrastructure grant for Hydroturbine design and test center.

Disclosure statement

No potential conflict of interest was reported by the author.

References

Akin, H., Aytac, Z., Ayancik, F., Ozkaya, E., Arioz, E., Celebioglu, K., & Aradag, S. (2013). A CFD aided hydraulic turbine design methodology applied to Francis turbines. IEEE 4th International Conference on Power Engineering, Energy and Electrical Drives (POWERENG).

Anup, K. C., Thapa, B., & Lee, Y. (2014). Transient numerical analysis of rotor–stator interaction in a Francis turbine. Renewable Energy, 65, 227–235.

Ardanuy, F., Wilhelmi, J. R., Mora, J. J., & Perez, L. (2006). Adjustable speed hydro generation: Operational aspects and control. IEEE Transactions on Energy Conversion, 21, c2–c2.

Ayancik, F., Celebioglu, K., & Aradag, S. (2014). Parametrical and theoretical design of a Francis turbine runner with the help of computational fluid dynamics. 10th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics, Orlando, FL.

Ayli, E., Kaplan, A., Cetinturk, H., Kavurmaci, B., Demirel, G., Celebioglu, K., & Aradag, S. (2015). CFD analysis of 3D flow for 1.4 MW Francis turbine and model turbine manufacturing. ASME 2015 International Design Engineering Technical Conferences and Computers and Information in Engineering Conference. doi:10.1115/DETC2015-46258.

Daneshkah, K., & Zangeneh, M. (2010). Parametric design of a Francis turbine runner by means of a three-dimensional inverse design method. 25th IAHR Symposium on Hydraulic Machinery Systems, Vol. 12.

Farell, C., Arroyave, J., Cruz, N., & Gulliver, J. S. (1993). Hydrodynamics of variable speed turbines (Project Report No 225). University of Minnesota St. Anthony Falls Hydraulic Laboratory, Department of Natural Resources.

Gjøsæter, K. (2011). Hydraulic design of Francis turbine exposed to sediment erosion (MS thesis). Norwegian University of Science and Technology, Department of Energy and Process Engineering.

Gohil, P. P., & Saini, R. P. (2014). CFD: Numerical analysis and performance prediction in Francis turbine. IEEE 1st International Conference on Non-Convention Energy (ICONCE).

Keck, H., & Sick, M. (2010). Thirty years of numerical flow simulation of Francis Turbines. Revista Facultad De Ingenieria Universidad De Antioquia, 51, 34–33.
Kurosawa, S., Lim, S. M., & Enomoto, Y. (2010). *Virtual model test for a Francis turbine*. IOP Conference Series: Earth and Environmental Science. Vol. 12(1). IOP Publishing.

Obrovsky, J., Krausova, H., Spidla, J., & Zouhar, J. (2013). Development of high specific speed Francis turbine for low head HPP. *Engineering Mechanics*, 20(2), 139–148.

Patel, K., Desai, J., Chauhan, V., & Charnia, S. (2011). *Development of Francis turbine using computational fluid dynamics*. 11st Asian International Conference on Fluid Machinery and 3rd Fluid Power Technology Exhibition, 1–3.

Qian, Z. D., Yang, J. D., & Huai, W. X. (2007). Numerical simulation and analysis of pressure pulsation in Francis hydraulic turbine with air admission. *Journal of Hydrodynamics, Ser. B*, 19(4), 467–472.

Round, G. F. (2004). *Incompressible flow turbomachines: Design, selection, applications and theory*, USA (pp. 8–24). Oxford: Gulf Professional Publishing.

Su, W. T., Li, F. C., Li, B. X., Wei, X. Z., & Zhao, Y. (2012). Assessment of LES performance in simulating complex 3D flows in turbo-machines. *Engineering Applications of Computational Fluid Mechanics*, 6(3), 356–365.

Zhang, H., & Zhang, L. (2012). Numerical simulation of cavitating turbulent flow in a high head Francis turbine at part load operation with OpenFOAM. *Procedia Engineering*, 31, 156–165.