Proceedings of COBEM 2011 21st Brazilian Congress of Mechanical Engineering
October 24-28, 2011, Natal, RN, Brazil

CONCEPTUAL DESIGN OPTIMIZATION OF FRANCIS TURBINES

Rodrigo Barbosa da Fonseca e Albuquerque, galegosigma@yahoo.com.br
Waldir de Oliveira, waldir@unifei.edu.br
Universidade Federal de Itajubá, Av. BPS 1303, C.P. 50, CEP 37500-903, Itajubá, MG, Brazil.

Abstract. *Before running high-cost CFD-based tools in the design process of turbomachines, it is advisable to evaluate first suitable configurations for the key geometric parameters of runners and stators. This can be achieved by coupling simplified models for the turbomachine fluid flow with numerical optimization techniques. This work presents such a conceptual design methodology for Francis turbines. First, the 1D flow solver is developed with the aim of easily identifying performance improvements in response to changes in the design variables. Then, optimization techniques – sequential quadratic programming, SQP, and controlled random search algorithm, CRSA – are presented and applied in order to find optimum design variables of a micro Francis turbine.*

Keywords*: hydraulic turbine, Francis turbine, mean streamline analysis, conceptual design optimization.*

1. INTRODUCTION

Hydraulic turbines are long history turbomachines. They have been systematically studied, designed, built and put into operation for nearly 250 years. Initially as a substitute for secular water wheels in mills, looms and small factories, hydraulic turbines are designed today, almost exclusively, to drive generators in hydroelectric power plants of all kinds.

In the early designs of hydraulic turbines, the experience of the engineers, along with numerous time consuming and expensive tests with scale models, many of whom carried out on a "trial-and-error" basis, were the main available design tools (Raabe, 1985; Ueda, 1982). Part of this accumulated empirical knowledge has been condensed into technical charts and diagrams (still in use today), which provide good guidelines for pre-dimensioning of turbines (Schweiger and Gregori, 1989; Lugaresi and Massa, 1987). Another part of this knowledge has been kept closed with the manufacturers, being passed from "hand-by-hand", as an inheritance, to the engineering teams of the companies. Actually, in comparison with other types of turbomachines, such as pumps, fans, compressors, gas- and steam-turbines, it can be said that there are relatively few open technical publications related to the design and optimization of hydraulic turbines.

The modern design of hydraulic turbines, as well as of virtually all turbomachines, is made with increasingly intense use of computer codes for numerical calculations of the fluid flow through the machine. These CFD (Computational Fluid Dynamics)-based tools are able to simulate accurately many important physical phenomena that occur in the flow inside the turbine, assisting the engineers in detailing the hydraulic profiles (Drtina and Sallaberger, 1999; Casey, 2003). Only after exhaustive numerical investigations the scale models are built for the final tests in test rigs. The time and total costs spent on developing new projects are significantly reduced with this systematics, and a high optimized performance can be achieved for the prototype turbine.

But although 3D Navier-Stokes codes have allowed good performance predictions and contributed for decreasing the costs of turbomachine model tests, a considerable computational effort has still to be spent with grid generation and with the solution of the flow governing equations in each numerical investigation. This issue is even more important in the case of design optimization: when a geometric change is made during the optimization process, complex meshes must be rebuilt and the flow solver must be run again. This high effort prevents the incorporation of sophisticated Navier-Stokes simulations in the whole design process (Hirsch and Demeulenaere, 2003). Actually, the analysis and design of turbomachines still require the use of simpler methodologies mainly in the preliminary design phases, when the geometry is not yet completely defined. One example of a very simple methodology is the mean streamline analysis for conceptual design optimization of pumps (Oh and Kim, 2001). For axial-flow gas turbines, it is common the use of the simplified radial equilibrium equation or streamline curvature methods for evaluating the radial flow variations (Kacker and Okapuu, 1982; Sullerey and Kumar, 1984).

For hydraulic turbines, however, the description of this kind of intermediate approach is apparently scarce in the open literature. On one hand, one can found theoretical analysis of the overall performance, which does not account for the effects of runner blade geometry changes in the flow field (Parker, 1996). On the other hand, it is not difficult to find modern design optimization strategies using direct CFD analysis, without previous systematic assessment on suitable geometric configurations (Kueny *et al.*, 2004; Lipej, 2004).

Therefore it seems desirable to make available intermediate-stage design tools for hydraulic turbines. These tools should provide a reliable conceptual design, i.e., with a simplified but representative geometry for runners and stators and also favorable trends towards the optimal flow field. This approach is performed for axial-flow water turbines in Albuquerque *et al.* (2007a) and for pump-turbines in Rodrigues dos Santos (2009). In the present work, this kind of methodology is proposed for Francis turbines.

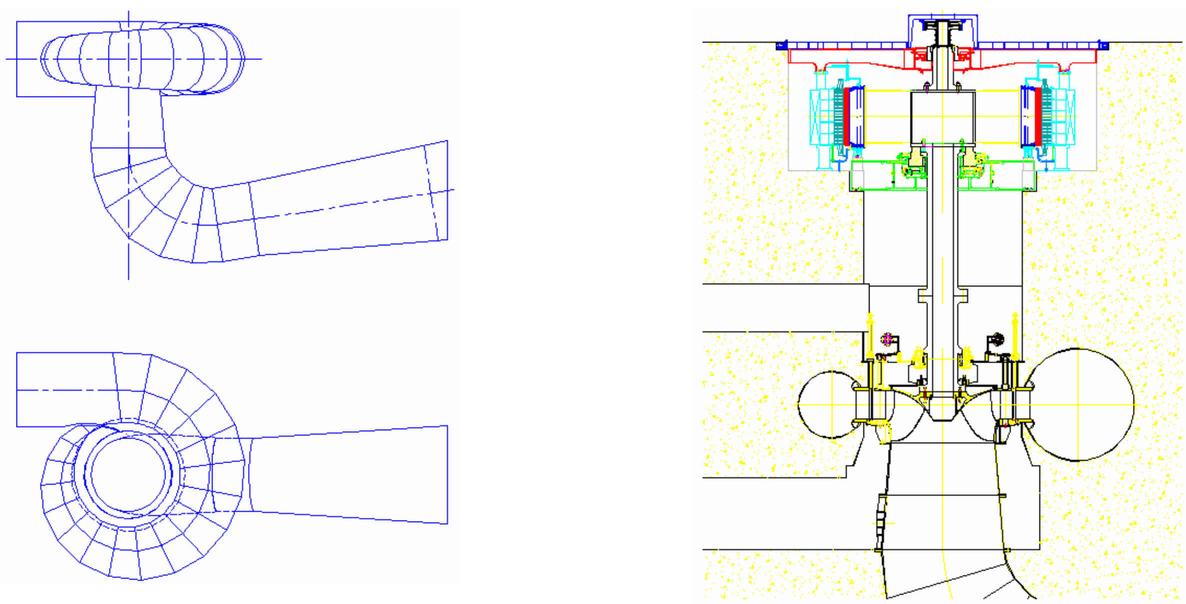
In this study, a low-cost flow solver is coupled with two different numerical optimization techniques. The flow solver is entirely based on the mean streamline analysis elaborated by Granja Jiménez (2004) for the theoretical prediction of some hydrodynamic characteristics of Francis turbines. In that work, the Euler equation for the turbomachines and various empirical losses correlations for each turbine component – taking in account only the main geometric parameters of the machine – are applied in order to evaluate the turbine efficiency hill-charts.

The design optimization problem is stated in Section 3. The proposed flow solver is discussed in Section 4. Section 5 shows the performance prediction of a given micro Francis turbine, by using the flow solver. A brief explanation about the chosen optimization methods is given in Section 6. In Section 7, the conceptual design optimization tool is applied and the results are compared with the Francis turbine design of Section 5. Concluding remarks are given in Section 8.

2. INDUSTRIAL IMPORTANCE OF THE FRANCIS TURBINES

Francis turbines, Fig. 1, are reaction-type hydraulic turbomachines, in which the working-fluid static pressure varies through the runner. This type of water turbine is of paramount importance in the overall context of hydropower due to their remarkable hydrodynamic and mechanical features, which allow its application in low head, medium head and high head exploitations, since small hydro to large hydropower plants (Quantz, 1976; Macintyre, 1983). Thus, Francis turbines compete with Kaplan and propeller turbines in the head range near 20 to 70 mWC (meters of water column), and with Pelton turbines in the head range near 200 to 800 mWC. In the medium head exploitations, with head range near 70 to 200 mWC, hardly any other type of water turbine exceeds the performance provided by Francis turbines. And this is precisely the head range in which lies the majority of the hydroelectric plants in operation worldwide, as well as a significant part of the new ones in study. Therefore, besides being the type of water turbine most found today, the Francis turbines will continue, even for a long time, being used in new hydropower plant projects, despite the strong growth of low and very low head power plants in the last decades – these ones using Kaplan and propeller turbines, many of them in bulb arrangements.

The specific speed n_{qA} ($=1000\times n(\mathrm{rps})\times Q(\mathrm{m^3/s})^{0.5}\times(g(\mathrm{m/s^2})\times H(\mathrm{m}))^{-0.75}$) practiced in Francis turbine designs typically varies from 70 to 450 (Macintyre, 1983; Voith, 2004). In terms of technology, we can say that the designs with n_{qA} from 150 to 300, approximately, are those ones more "mature", because they have been the most commonly used over time. The development now focuses most on the borders, i.e., in high head machines (low n_{qA}), improving competition with Pelton turbines, and, mainly, in low head machines (high n_{qA}), entering with even better performance in the field of axial turbines (Kaplan/propeller). This broad range of application, combined with good efficiency, straightforward mechanical design, competitive costs, good features of regulating, high strength, durability and low maintenance, make the Francis turbines the most used water-drive motors in micro, small, medium and large hydropower plants, and the main hydroelectric plants of the world are predominantly equipped with Francis turbines: Itaipu (14000 MW), on the border between Brazil and Paraguay; Belo Monte (11000 MW, in construction), Tucuruí (8125 MW), Ilha Solteira (3444 MW), Xingó (3162 MW), Paulo Afonso IV (2460 MW), Itumbiara (2082 MW), Foz do Areia (1676 MW), São Simão (1608 MW), Itaparica (1500 MW), Marimbondo (1440 MW), Salto Santiago (1420 MW), Água Vermelha (1396 MW), Serra da Mesa (1275 MW), Furnas (1216 MW), all in Brazil; Guri (10235 MW), in Venezuela; Three Gorges (17680 MW), in China; Grand Coulee (6809 MW), in United States of America; Churchill Falls (5428 MW), in Canada; Sayano-Shushenskaya (6400 MW) in Russia, among many other ones.



(a) (b)

Figure 1. Basic design of a large vertical-shaft Francis turbine with spiral case. (a) hydraulic lay-out (b) cross-section of the generating unit. (Courtesy of Alstom Power)

3. DESIGN OPTIMIZATION PROBLEM

In the present study, the Francis turbine design problem consists in searching some basic geometric parameters of the machine – the design variables – in order to maximize its efficiency – the objective function –, given the turbine rotational speed and the discharge. The turbine net head should lie within upper and lower limits, these being the nonlinear constraints of the problem. There are also lateral constraints for the design variables, defining the problem design space. Despite we are dealing with a single point optimization only, this can be useful for achieving also a good operating range (Ueda, 1982; Souza Júnior *et al.*, 2005).

Formally, the design problem can be stated as a constrained nonlinear minimization problem as follows:

minimize $f(\mathbf{x})$
subject to $g_i(\mathbf{x}) \leqslant 0$, $i = 1, \ldots, m.$
$\mathbf{x} \in S$

where $\mathbf{x}$ is the n-dimensional vector of the design variables, $\mathbf{x} = (x_1, x_2, \ldots, x_n)$. These design variables are some key geometric parameters of the turbine, defined in Section 7. The search region S is defined by upper and lower bounds, x_j^U and x_j^L respectively, for each coordinate of $\mathbf{x}$: $S = \{\mathbf{x} \in \Re^n : x_j^L \leq x_j \leq x_j^U, j = 1, \ldots, n\}$. The objective function is $f(\mathbf{x}) = -\eta(\mathbf{x})$, where η is the turbine efficiency (with the geometry according to $\mathbf{x}$). $g_i(\mathbf{x})$, $i = 1, \ldots, m$, are the $m = 2$ nonlinear constraint functions, namely, $g_1(\mathbf{x}) = H_L - H(\mathbf{x})$ and $g_2(\mathbf{x}) = H(\mathbf{x}) - H_U$, where H is the turbine net head and H_L and H_U are respectively lower and upper limits, such that $H_L \leqslant H \leqslant H_U$. The performance of the turbine (η and H) is evaluated by the flow solver, as described in Section 4.

The problem stated above can be solved by optimization algorithms that treat directly nonlinear constraints. Otherwise, the constraints for the net head can be imposed by means of a penalization scheme on the objective function:

$$f = \begin{cases} -\eta + M(H_L - H)^2 , & H < H_L \\ -\eta , & H_L \leq H \leq H_U \\ -\eta + M(H - H_U)^2 , & H > H_U \end{cases}$$

where M is a sufficiently large positive number. Again, the objective is to maximize η (minimize $-\eta$) with $H_L \leqslant H \leqslant H_U$. The choice of the penalty factor M must not drive the optimization process towards a penalty minimization only, missing the objective function main information, i.e., the turbine efficiency, η. Also, the constraints must not be violated at the end of the process. Some tests have to be performed in order to settle suitable values for M in each particular problem.

4. FLOW SOLVER

Despite of being a very simple approach, the mean streamline (or 1D) analysis is a useful way for evaluating the overall performance of turbomachines in the beginning of the design conception. For example, Kacker and Okapuu (1982) coded a mean line method for the efficiency prediction of axial-flow gas turbines. Oh and Kim (2001) coupled a gradient-based search algorithm with a mean streamline system for the conceptual design optimization of mixed-flow pump impellers.

In water turbines, the mean line approach is also a suitable way for starting the design process (Souza, 1991; Macintyre, 1983). In the work of Granja Jiménez (2004), a mean line analysis system was developed and coded in MatLab™ language for the theoretical performance prediction of Francis turbines. The predicted hill-charts of a specific Francis turbine design fitted well the measured data on the corresponding model test. The hydraulic losses were evaluated by using empirical correlations given in various references. These losses correlations make use of empirical factors, flow velocities, flow angles and geometric quantities; all of them are taken in the mean streamline, as average values for each turbine section.

Therefore, for the purposes of the present study, the performance calculation code of Granja Jiménez (2004) was adapted for being the Francis turbine flow solver. In a further advanced design stage, the mean line analysis would not be enough for refining the machine geometry, and thus, one should apply a 2D- or even a 3D-approach in that time – but following the optimum conceptual design already obtained.

The Francis turbine lay-out considered in this work is the classical vertical-shaft Francis turbine, with spiral case and elbow-type draft tube (Fig. 1). The main hydraulic components of the turbine are the spiral case, the stay vanes, the wicket gates (guide vanes), the runner and the draft tube, Fig. 2.

The following assumptions are made for the flow calculation: 1) incompressible fluid flow; 2) stationary absolute flow in the spiral case, stay vanes, wicket gates and draft tube; 3) stationary relative flow in the runner; 4) stationary flow in the space between the stay vanes and the wicket gates and between the wicket gates and the runner; 5) the absolute flow (in the stationary components) or the relative flow (in the runner) occur in revolution surfaces, concentric with the turbine shaft; 6) the absolute flow (in the stationary components) or the relative flow (in the runner) are uniform in the sections upstream and downstream of each component; 7) the flow quantities of interest are evaluated in the mean streamsurfaces (absolute flow for the stationary components, and relative flow for the runner), in the meridional plane, for each component; 8) the inlet and outlet points in the mean streamlines (inlet and outlet diameters) of each component in the meridional plane are defined by the continuity equation.

In the work of Granja Jiménez (2004), the flow velocities and the flow angles are calculated at the inlet and outlet sections of each Francis turbine component, on the mean streamline. These quantities are necessary for the evaluation of the hydraulic losses in each component and also for the torque calculation on the runner blades. For this aim, Granja Jiménez (2004) considered the key geometric parameters of each component and also the continuity equation for a given discharge Q. Some correlations were also used for the evaluation of the outlet flow deviation angles in the stay vanes, wicket gates and runner blades. For the vaneless spaces between the stay vanes and the guide vanes and between the guide vanes and the runner, some corrections for the usual free-vortex hypothesis were also applied. The following figures show the main geometric parameters of each component and the corresponding velocity diagrams. All the details of the flow velocities and flow angles calculations are minutely given in the work of Granja Jiménez (2004).

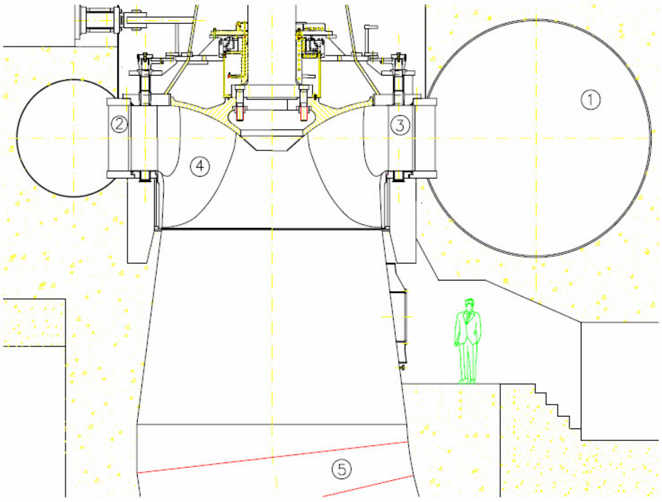


Figure 2. Basic hydraulic components of a vertical-shaft Francis turbine: 1) spiral case; 2) stay vanes; 3) wicket gates; 4) runner; 5) draft tube. (Courtesy of Alstom Power)

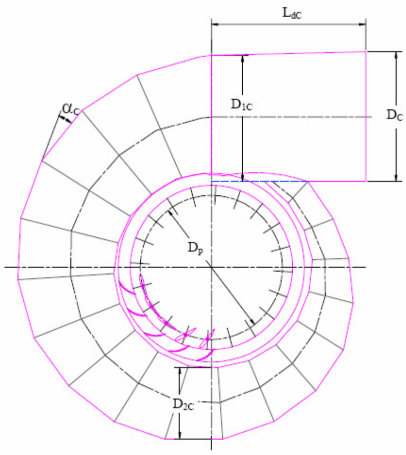


Figure 3. Main geometric parameters of the Francis turbine spiral case. (Granja Jiménez, 2004)

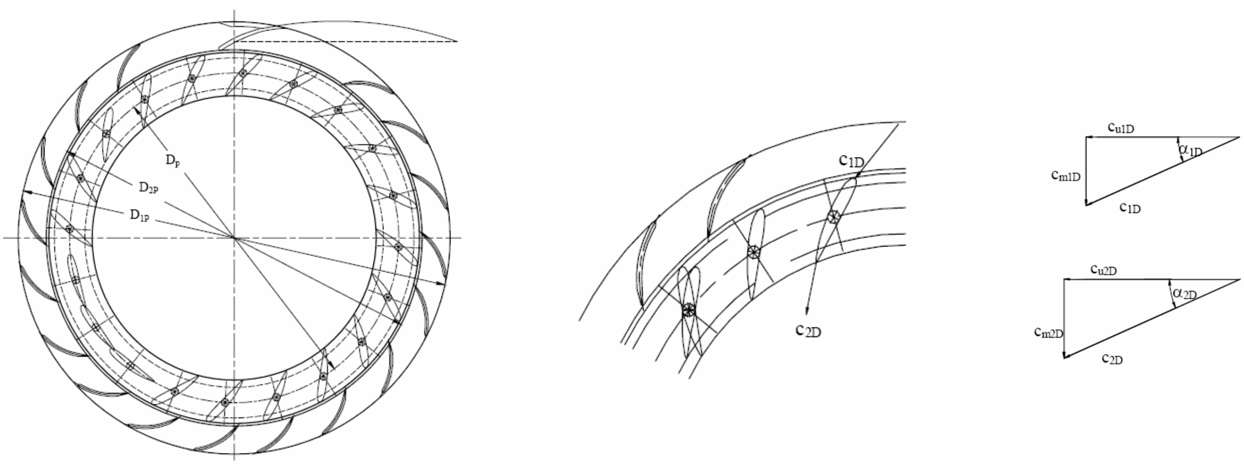


Figure 4. Main geometric parameters of the Francis turbine stay vanes/wicket gates and velocity diagrams. (Granja Jiménez, 2004)

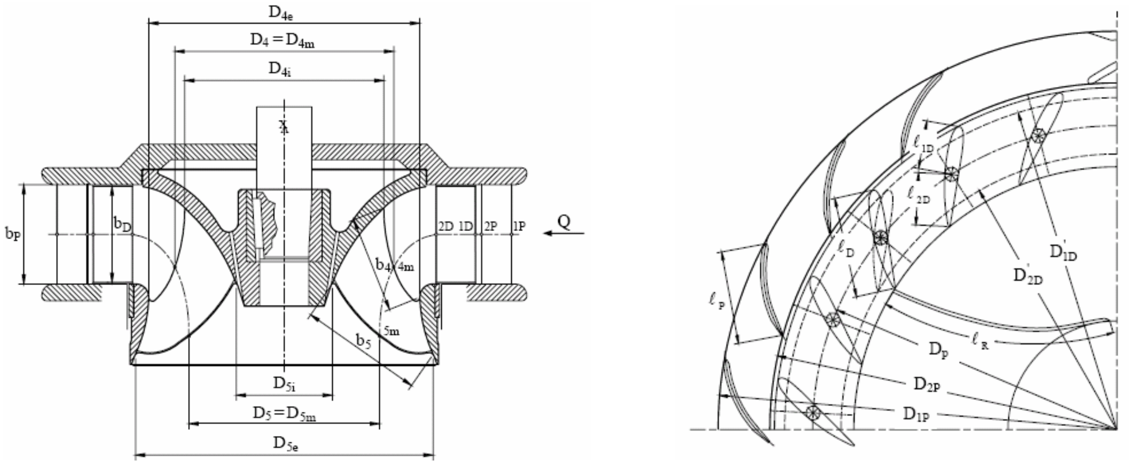


Figure 5. Main geometric parameters of the Francis turbine runner. (Granja Jiménez, 2004)

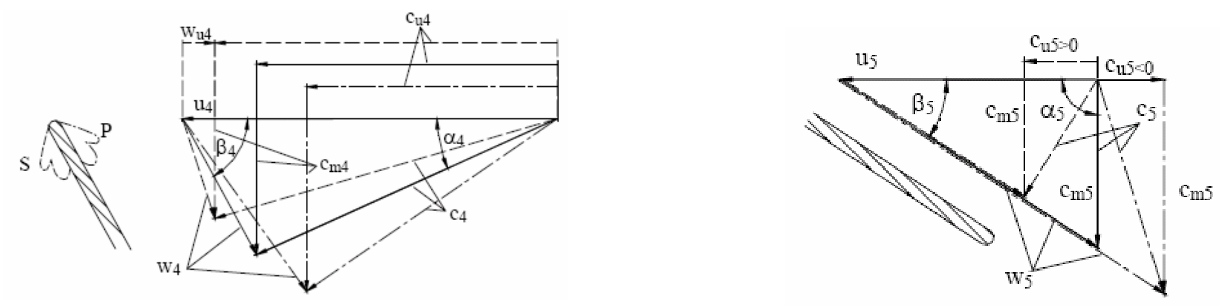


Figure 6. Velocity triangles at the Francis turbine runner blades. (Granja Jiménez, 2004)

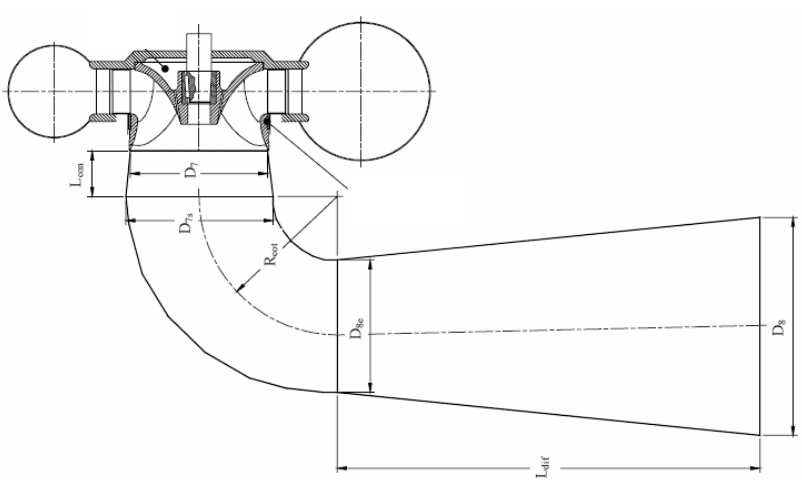


Figure 7. Main geometric parameters of the Francis turbine draft tube. (Granja Jiménez, 2004)

The main hydraulic losses in each turbine component are evaluated by using various empirical correlations given in the technical literature (Granja Jiménez, 2004). Table 1 summarizes the loss mechanisms elected in this analysis.

The hydraulic losses (due to friction) are also evaluated in the vaneless spaces between the stay vanes and the wicket gates and between the wicket gates and the runner.

In addition to the hydraulic losses, it is also computed: 1) the leakage loss, due to the leakage flow through the labyrinths between the runner wearing plates and the covers; 2) the disk friction loss between the runner crown and the head cover and between the runner band and the bottom ring; 3) the mechanical (or external) losses, due to mechanical friction in the turbine bearing(s) and seals. We denote as Z_h the total hydraulic loss through the turbine; the total disk friction loss is denoted by Z_{df} and the total leakage loss is denoted by Z_{leak}. These losses are given in terms of energy per unit weight. The mechanical loss, P_{mech}, is treated directly as a loss of power in the turbine shaft.

Table 1. Hydraulic losses considered in the Francis turbine components.

Turbine component	*Hydraulic loss mechanisms*
Spiral case	Friction, bend
Stay vanes	Incidence, friction, wake mixing
Wicket gates	Incidence, friction, wake mixing
Runner	Incidence, friction, wake mixing
Draft tube cone	Swirl flow, diffuser flow
Draft tube elbow	Friction, bend
Draft tube diffuser	Diffuser flow

As previously mentioned, these losses are evaluated by using empirical correlations. The calculation of all these losses is detailed in the work of Granja Jiménez (2004). The empirical factors, as well as the other necessary geometric and kinematic quantities, are clearly described in that work.

It is important to bear in mind that the set of losses correlations in the work of Granja Jiménez (2004) was chosen with the aim of covering the main sources of losses in Francis turbines. For conceptual design optimization purposes, the level of accuracy of the losses predictions is not the main concern. The key point is the capability of the loss modeling in indicating the correct trends of losses variations due to geometry changes, so that different designs can be judged in a comparative sense (Casey, 2003; Denton, 1993). The set of loss models should drive the search towards a geometric configuration that yields suitable hydrodynamic features.

The theoretical specific work absorbed by the runner blades, $H_{\text{blade\_th}}$ (in terms of energy per unit weight), is given by the Euler classic equation for the turbines (Quantz, 1976), that represents the integral angular momentum equation:

$$H_{\text{blade\_th}} = (u_4 c_{u4} - u_5 c_{u5}) / g \tag{1}$$

where u is the runner peripheral velocity; c_u is the circumferential component of the absolute flow velocity; the index 4 denotes the runner inlet section; the index 5 denotes the runner outlet section; g is the acceleration due to gravity. The quantities are all considered on the mean streamline, as previously mentioned.

In the analysis of Granja Jiménez (2004), the actual specific work absorbed by the runner blades, H_{blade}, is evaluated according to the method proposed by Pfleiderer and Petermann (1972):

$$H_{\text{blade}} = H_{\text{blade\_th}} / (1 + p) \tag{2}$$

where p is a semi-empirical factor which takes in account the number of runner blades and also some geometric parameters of the runner.

The turbine net head, H, – i.e., the energy per unit weight provided by the water flow – is calculated by:

$$H = H_{\text{blade}} + Z_{\text{h}} \tag{3}$$

The corresponding available power, P_{avail}, is given by:

$$P_{\text{avail}} = \rho Q g H \tag{4}$$

where ρ is the water density.

The shaft power output, P_{shaft}, is calculated by:

$$P_{\text{shaft}} = \rho Q g (H_{\text{blade}} - Z_{\text{leak}} - Z_{\text{df}}) - P_{\text{mech}} \tag{5}$$

Finally, the turbine total efficiency, η, is calculated by:

$$\eta = P_{\text{shaft}} / P_{\text{avail}} \tag{6}$$

The flow solver is a computer program in MatLab™ language for performing the calculation sequence sketched above. Given the geometric parameters – i.e., the turbine conceptual design –, the rotational speed, n, and the discharge, Q, the flow solver calculates the flow velocities, the flow angles, the losses, the net head, the shaft power and the total efficiency at this operating point. So, this flow solver is suitable for the proposed design optimization system: the optimization algorithm provides the flow solver the turbine design variables, $\mathbf{x}$, to be evaluated; the flow solver calculates the net head, H, and the total efficiency, η, as output results (n and Q are pre-defined); the optimization algorithm evaluates these results and then changes the design variables $\mathbf{x}$ (in the design space S) trying to maximize η and to keep H within the constraints. The process runs until the convergence criteria are reached or the maximum allowable number of iterations is exceeded.

The empirical factors values in the present flow solver – for the evaluation of the outlet flow angles deviations, the losses and the free-vortex corrections in the vaneless spaces – are the same ones used by Granja Jiménez (2004).

5. EXAMPLE OF CALCULATION WITH THE FLOW SOLVER

The flow solver just described was applied for the performance evaluation of a micro Francis turbine with n_{qA} = 266. The main geometric parameters of this turbine are given in Tab. 2 (see also Figs. 3 to 7).

Table 2. Main geometric parameters of the studied Francis turbine.

Spiral case		***Wicket gates***	
Inlet diameter, D_C	314.3 mm	Number of guide vanes, N_{pd}	20
Inlet length, L_{inj}	380.0 mm	Pitch diameter, D_p	347.2 mm
Diameter downstream the reduction, D_{1C}	304.5 mm	Chord length, ℓ_D	62.4 mm
Diameter at 180 degrees, D_{2C}	188.0 mm	Chord length upstream the trunnion, ℓ_{1D}	30.0 mm
Linear length, L_C	1501 mm	Camber, f	0.6 mm
Spiral angle, α_C	14°	Height (constant), b_D	98.0 mm
Stay vanes		***Runner***	
Number of stay vanes, N_{pf}	18	Number of blades, N_R	12
Inlet diameter, D_{1P}	456.1 mm	Inlet diameter, D_4	218 mm
Outlet diameter, D_{2P}	397.2 mm	Outlet diameter, D_5	180 mm
Chord length, ℓ_P	61.0 mm	Chord length, ℓ_R	90 mm
Inlet angle, α_{1P}	14°	Inlet angle, β_4	52°
Outlet angle, α_{2P}	45°	Outlet angle, β_5	26°
Inlet height, b_{1P}	98.6 mm	Inlet height, b_4	86 mm
Outlet height, b_{2P}	98.6 mm	Outlet height, b_5	110 mm
Draft tube		Elbow outlet diameter, D_{8e}	284.7 mm
Cone inlet diameter, D_7	307.5 mm	Elbow radius, R_{cot}	277.8 mm
Cone outlet diameter, D_{7s}	326.4 mm	Diffuser length, L_{dif}	1194 mm
Cone length, L_{con}	100.0 mm	Diffuser outlet diameter, D_8	535.3 mm

The rated operating conditions of this turbine are given in Tab. 3. The wicket gates opening, *a*, that corresponds to this operating point is *a* = 30.1°. By applying the flow solver, some hydrodynamic characteristics of interest were evaluated at this operating point, as shown in Tab. 4.

Table 3. Design point of the studied Francis turbine.

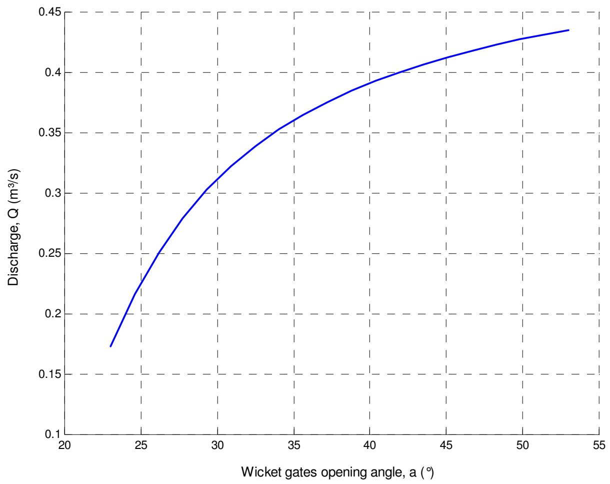
Net head, *H*	12.6 mWC
Discharge, *Q*	0.313 m³/s
Rotational speed, *n*	1130 rpm

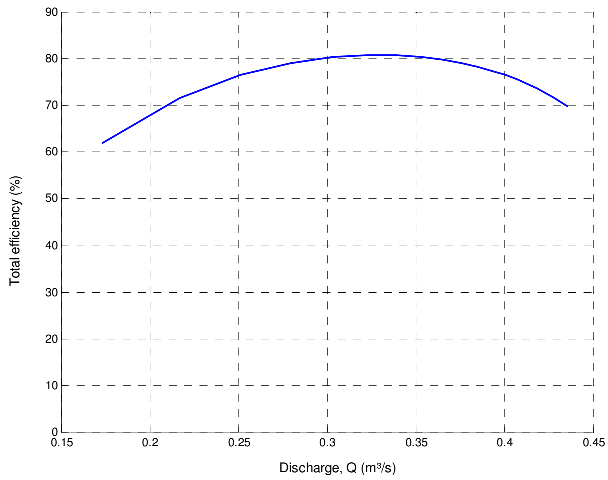
Table 4. Some design point hydrodynamic characteristics of the studied Francis turbine.

Hydraulic losses in the spiral case, Z_{hC}	0.21 mWC
Hydraulic losses in the stay vanes, Z_{hP}	0.12 mWC
Hydraulic losses in the wicket gates, Z_{hD}	0.70 mWC
Hydraulic losses in the runner, Z_{hR}	0.36 mWC
Hydraulic losses in the draft tube, Z_{hDT}	0.49 mWC
Total hydraulic losses in the turbine, Z_h	1.88 mWC
Runner blades specific work, H_{blade}	10.7 mWC
Shaft power output, P_{shaft}	31.1 kW
Turbine total efficiency, η	80.64 %

For the $\eta \times Q$ performance prediction (keeping constant *n* and *H*), an iterative calculation scheme should be performed. For each wicket gates opening *a*, within the opening range, it should be calculated the discharge *Q* that corresponds to this opening with the given net head and rotational speed, according to the flow solver. So, for each wicket gates opening, an implicit nonlinear problem has to be solved for the discharge. This is carried out by coupling (for each step of the wicket gates opening) the flow solver with the `fzero` function from MatLab™ toolbox. This M-function uses an efficient combination of bisection, secant, and inverse quadratic interpolation methods, being suitable for the present problem.

So, for $23° \leqslant a \leqslant 53°$, the $\eta \times Q$ performance prediction (keeping *n* = 1130 rpm and *H* = 12.6 mWC) is given in Fig. 8b. Figure 8a shows the calculated $Q \times a$.





(a) (b)

Figure 8. (a) discharge *versus* wicket gates opening angle and (b) theoretical efficiency curve of the studied Francis turbine. (n = 1130 rpm ; H = 12.6 mWC)

6. OPTIMIZATION METHODS

The conceptual design optimization system is completed by coupling the flow solver with suitable optimization techniques. In the present study, two optimization methods have been alternatively tested for this aim: a Sequential Quadratic Programming method (SQP) and a Controlled Random Search Algorithm (CRSA). The SQP is a gradient-based method, being useful for local searches starting from previous designs. The CRSA is a population set-based direct search algorithm that helps in exploratory searching throughout the design space.

The SQP is one of the most efficient local optimization methods for solving constrained nonlinear problems (Nash and Sofer, 1996), being suitable for the present application. In this study, the `fmincon` function from MatLab™ optimization toolbox was chosen for performing the SQP. This is an efficient implementation of the standard SQP using the BFGS formula for approximating the Hessian matrix (Nash and Sofer, 1996). In the present application, the option for evaluating the directional derivatives by finite differences was set.

The two main drawbacks of a gradient-based method are the search only for local optimizers and the need of a starting point for the design variables. The success of the search may become very dependent of this starting guess and thus requires that the engineer provides an initial design not too far from an acceptable optimum.

To try overcoming these limitations, the CRSA has also been applied in this study. The CRSA was first proposed by Price (1977) and substantially improved by Ali *et al.* (1997a). Like genetic and differential evolution algorithms, the CRSA is a population set-based global algorithm. It starts with an initial population of points on the design space and then performs iterative substitutions of worst points by better points in order to contract the whole population towards a global optimizer. In CRSA, a single point is replaced per iteration. The CRSA was chosen due to its straightforward implementation, fastness and good results reported in technical literature (Ali *et al.*, 1997b; Manzanares-Filho *et al.*, 2005). Here one applies the algorithm proposed by Ali *et al.* (1997a) with modifications for avoiding ill-conditioning and accelerating the convergence when solutions lie in the vicinity of the design space boundaries (Albuquerque, 2006; Albuquerque *et al.*, 2007b).

Differently of the SQP, the CRSA does not require a careful starting design. CRSA employs an initial population randomly chosen on the design space *S*. Besides to alleviate the designer's effort, the CRSA increases the hope of finding a global optimum. A relatively small number of function evaluations for convergence is also an important feature of CRSA (Ali and Törn, 2004; Albuquerque *et al.*, 2007b).

When using the CRSA, the net head constraints are imposed by means of the penalty scheme on the objective function as described in Section 3.

7. RESULTS

The micro Francis turbine studied in Section 5 has been used for an application example of the proposed conceptual design methodology. Five geometric parameters have been chosen as the design variables: the stay vanes inlet and outlet angles (α_{1P} and α_{2P}), the wicket gates opening angle (a) and the runner blades inlet and outlet angles (β_4 and β_5). These quantities have been chosen in order to easily identify performance improvements at this intermediate design stage. The optimization is related only to the rated operating point (Tab. 3). The other geometric data of the turbine (Tab. 2) were keep unmodified.

The optimization runs were performed according to Table 5 for the design variables constraints. The best results are compared with the basic design in Table 6.

Table 5. Design variables lateral constraints (search region *S*).

Design variable	*Lower bound*	*Upper bound*
stay vanes inlet angle, α_{1P}	10°	35°
stay vanes outlet angle, α_{2P}	25°	55°
wicket gates opening angle, *a*	25°	55°
runner blades inlet angle, β_4	40°	70°
runner blades outlet angle, β_5	15°	35°

Table 6. Comparison of the optimization results with the basic design of the studied Francis turbine.

Design variables and characteristics	***Basic** design*	*Best **SQP** solution found*	*Best **CRSA** solution found*
stay vanes inlet angle, α_{1P} (°)	14	29.5	29.4
stay vanes outlet angle, α_{2P} (°)	45	25.1	25.1
wicket gates opening angle, *a* (°)	30.1	30.5	30.4
runner blades inlet angle, β_4 (°)	52	52.0	56.3
runner blades outlet angle, β_5 (°)	26	24.6	24.7
Hydraulic losses in the spiral case, Z_{hC} (mWC)	0.21	0.21	0.21
Hydraulic losses in the stay vanes, Z_{hP} (mWC)	0.12	0.03	0.03
Hydraulic losses in the wicket gates, Z_{hD} (mWC)	0.70	0.31	0.32
Hydraulic losses in the runner, Z_{hR} (mWC)	0.36	0.38	0.38
Hydraulic losses in the draft tube, Z_{hDT} (mWC)	0.49	0.48	0.48
Total hydraulic losses in the turbine, Z_h (mWC)	1.88	1.41	1.41
Runner blades specific work, H_{blade} (mWC)	10.7	11.2	11.2
Shaft power output, P_{shaft} (kW)	31.1	32.4	32.4
Turbine total efficiency, η (%)	80.64	84.21	84.20

Several starting points were tried for the SQP method and most of the runs converged nearly for the same point, with slight coordinate differences. Curiously, the best SQP solution found was achieved when starting from the basic design. The SQP convergence processes required an average number of function evaluations (solver calls) of near 300.

The CRSA method was also run several times, each one using a distinct initial population of points (designs) randomly chosen in the design space (Tab. 5). Again, most of the solutions converged nearly for the same point, despite now a higher dispersion in the points coordinates. The CRSA contraction processes required an average number of function evaluations (solver calls) of near 1500.

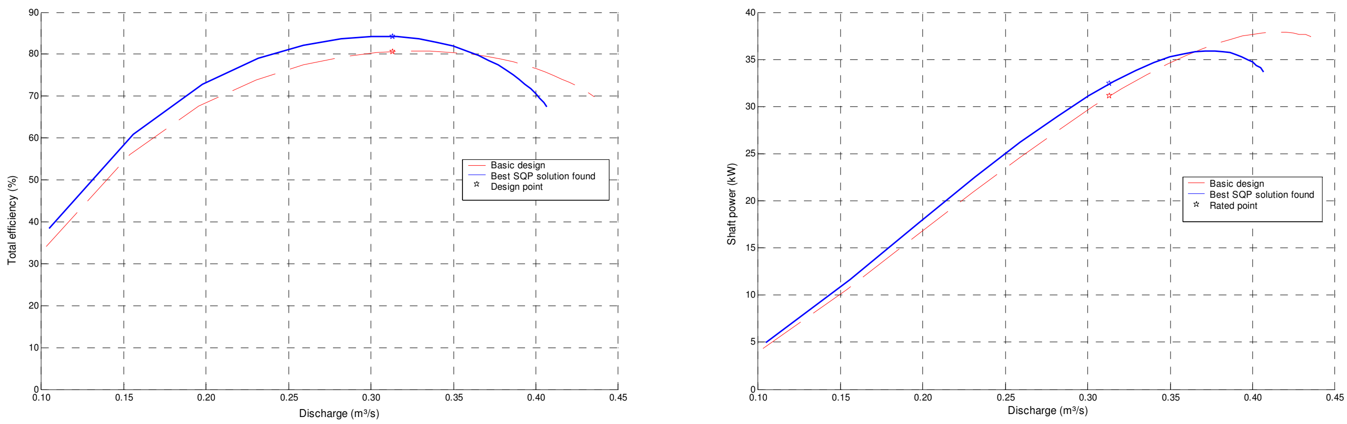
The results of SQP and CRSA optimization methods have shown good agreement in this problem. Apparently, the "global optimum" conceptual design – according to the turbine flow modeling – was found. Furthermore, the solution space in this problem seems to be very concave, i.e., with only one local minimum; thus, a (fast) gradient-based search method is as effective as a global one.

In the basic design, the flow solver analysis shows relatively high hydraulic losses in the stay vanes (0.12 mWC) and also in the wicket gates (0.70 mWC); in this last component, the hydraulic losses were even higher than those ones in the runner (0.36 mWC) or in the draft tube (0.49 mWC), what is not common in Francis turbines (Quantz, 1976; Ueda, 1982). Furthermore, the basic design stay vanes present a relatively high camber (see the great difference between α_{1P} and α_{2P}), what is unusual in Francis turbines. Therefore, the performance improvements achieved by the optimization system are due mainly to changes in the stay vanes inlet and outlet angles, which were substantially modified in comparison with the basic design ones, while the remaining design variables had slight modifications (Tab. 6). With these changes, the stay vanes became straighter (see the small difference between α_{1P} and α_{2P}), as the usual design practice in medium specific speed Francis turbines. These new stay vanes are more adapted for receiving the water flow from the spiral case, reducing the stay vanes incidence loss; the stay vanes total hydraulic losses were reduced to only 0.03 mWC. Further, the optimized stay vanes exit flow is more favorable for the wicket gates, with great reduction in the incidence loss and, thus, in its total hydraulic losses (reduced to only 0.31 mWC). Since the total hydraulic losses were reduced (to 1.41 mWC), the runner blades can absorb more energy and the runner specific work increases (to 11.2 mWC). The overall result is a higher total efficiency (84.21%) and a higher shaft power output (32.4 kW).

Despite we had carried out a single point optimization – at the rated operating point of Tab. 3 –, the performance has improved also in a reasonable range around this point, as shown in Fig. 9. Indeed, this is a well known feature for hydraulic turbines (Ueda, 1982).

However, the performance prediction makes sense only in the validity range of the used empirical correlations and simplifying assumptions. The results in very off-design conditions must be analyzed carefully.

Again, it must be stressed that the key point in this conceptual design methodology is the capability of the flow modeling in indicating the correct trends of performance variations due to changes in the design variables, so that different designs can be judged in a comparative sense.



(a) (b)

Figure 9. (a) efficiency *versus* discharge and (b) shaft power *versus* discharge for the basic and optimized designs. (n = 1130 rpm ; H = 12.6 mWC)

8. CONCLUDING REMARKS

A conceptual design optimization system has been proposed for Francis turbines. The flow solver is a low-cost computer code based on a mean streamline analysis. The losses are evaluated by using various empirical correlations given in the literature.

Two different optimization techniques have been alternatively coupled with the flow solver for the automatic searching for optimum turbine designs. One method is the sequential quadratic programming (SQP) – a gradient-based local search algorithm – and the other one is the controlled random search algorithm (CRSA) – a population set-based global search algorithm.

First, the performance of a micro Francis turbine was evaluated by using the flow solver. Then, the optimization system carried out searches for the optimum conceptual design, given the set of design variables and the operating point. The results were compared with the basic Francis turbine design showing potential performance improvements.

9. ACKNOWLEDGEMENTS

The authors wish to thanks to M.Sc. Eng. Ruth Katiuska Granja Jiménez for sending generously the computer programs that were the basis for the flow solver code in the present work.

10. REFERENCES

Albuquerque, R.B.F., 2006, “Design of Axial-Flow Hydraulic Turbines with Geometry Parameterization, the Radial Equilibrium Equation and Optimization Techniques” (in Portuguese), MSc Dissertation, Universidade Federal de Itajubá, Itajubá, MG, Brazil, 94 p.

Albuquerque, R.B.F., Manzanares-Filho, N. and Oliveira, W., 2007a, “Conceptual optimization of axial-flow hydraulic turbines with non-free vortex design”, Proc. Instn. Mech. Engrs, Part A, Journal of Power and Energy, Vol. 221, pp. 713-725.

Albuquerque, R.B.F., Manzanares-Filho, N. and Sousa, B.S., 2007b, “A Comparative Study of Controlled Random Search Algorithms with Application to Inverse Airfoil Design”, 28th Iberian Latin American Congress on Computational Methods in Engineering, Porto, Portugal, 19 p.

Ali, M.M., Törn, A. and Viitanen, S., 1997a, “A Numerical Comparison of Some Modified Controlled Random Search Algorithms”, Journal of Global Optimization, Vol. 11, pp. 377-385.

Ali, M.M., Storey, C. and Törn, A., 1997b, “Application of some stochastic global optimization algorithms to practical problems”, Journal of Optimization Theory and Applications, Vol. 95, pp. 545-563.

Ali, M.M. and Törn, A., 2004, “Population set-based global optimization algorithms: some modifications and numerical studies”, Computers & Operations Research, Vol. 31, pp. 1703-1725.

Casey, M., 2003, “Best Practice Advice for CFD in Turbomachinery Design”, QNET-CFD Network Newsletter, in A Thematic Network for Quality and Trust in the Industrial Application of CFD, Vol. 2, pp. 35-37.

Drtina, P. and Sallaberger, M., 1999, “Hydraulic turbines – basic principles and state-of-the-art computational fluid dynamics applications”, Proc. Instn. Mech. Engrs., Part C, Journal of Mechanical Engineering Science, Vol. 213, pp. 85-102.

Denton, J.D., 1993, “Loss Mechanisms in Turbomachines”, ASME Journal of Turbomachinery, Vol. 115, pp. 621-656.

Granja Jiménez, R.K., 2004, “Theoretical prediction of the hydrodynamic characteristics of Francis turbines” (in Portuguese), MSc Dissertation, Universidade Federal de Itajubá, Itajubá, MG, Brazil, 163 p.

Hirsch, C. and Demeulenaere, A., 2003, “State of the Art in the Industrial CFD for Turbomachinery Flows”, QNET-CFD Network Newsletter, in A Thematic Network for Quality and Trust in the Industrial Application of CFD, Vol. 2, pp. 5–9.

Kacker, S.C. and Okapuu, U., 1982, “A Mean Line Prediction Method for Axial Flow Turbine Efficiency”, ASME Journal of Engineering for Power, Vol. 104, pp. 111-119.
Kueny, J.-L., Lestriez, R., Helali, A., Demeulenaere, A. and Hirsch, C., 2004, “Optimal design of a small hydraulic turbine”, Proceedings of 22nd IAHR Symposium on Hydraulic Machinery and Systems, Stockholm, paper A02-2.
Lipej, A., 2004, “Optimization method for the design of axial hydraulic turbines”, Proc. Instn. Mech. Engrs, Part A, Journal of Power and Energy, Vol. 218, pp. 43-50.
Lugaresi, A. and Massa, A., 1987, “Designing Francis turbine: trends in the last decade”, Water Power & Dam Construction, November, pp. 23-28.
Macintyre, A.J., 1983, “Máquinas Motrizes Hidráulicas”, Guanabara Dois, Rio de Janeiro, Brazil, 649 p.
Manzanares-Filho, N., Moino, C.A.A. and Jorge, A.B., 2005, “An Improved Controlled Random Search Algorithm for Inverse Airfoil Cascade Design”, 6th World Congresses of Structural and Multidisciplinary Optimization, paper Nr. 4451.
Nash, S.G. and Sofer, A., 1996, “Linear and Nonlinear Programming”, McGraw-Hill, New York, United States of America, 692 p.
Oh, H.W. and Kim, K.-Y., 2001, “Conceptual design optimization of mixed-flow pump impellers using mean streamline analysis”, Proc. Instn. Mech. Engrs, Part A, Journal of Power and Energy, Vol. 215, pp. 133-138.
Parker, G.J., 1996, “A theoretical study of the performance of an axial flow turbine for a microhydro installation”, Proc. Instn. Mech. Engrs, Part A, Journal of Power and Energy, Vol. 210, pp. 121-129.
Pfleiderer, C. and Petermann, H., 1972, “Strömungsmaschinen”, Springer Verlag, Berlin, Germany, 550 p.
Price, W.L., 1977, “A controlled random search procedure for global optimisation”, Computer Journal, Vol. 20, Nr. 4, pp. 367-370.
Quantz, L., 1976, “Motores hidráulicos”, Editorial Gustavo Gili, Barcelona, Spain, 231 p.
Raabe, J., 1985, “Hydro power – The design, use, and function of hydromechanical, hydraulic, and electrical equipment”, VDI-Verlag GmbH, Düsseldorf, Germany, 684 p.
Rodrigues dos Santos, M.A., 2009, “Optimization of pump-turbine by using sequential quadratic programming and genetic algorithms” (in Portuguese), MSc Dissertation, Universidade Federal de Itajubá, Itajubá, MG, Brazil, 171 p.
Schweiger, F. and Gregori, J., 1989, “Developments in the design of water turbines”, Water Power & Dam Construction, May, pp. 23-26.
Souza Júnior, F., Silva, R.J., Manzanares-Filho, N., Barbosa, J.R. and Tomita, J.T., 2005, “Design Point Efficiency Optimization of a Multi-Stage Axial-Flow Compressor for Aero Application applying a Specially Developed Computer Code”, Proceedings of the 18th International Congress of Mechanical Engineering, Ouro Preto, Brazil, paper Nr. 1802.
Souza, Z., 1991, “Dimensionamento de máquinas de fluxo: turbinas-bombas-ventiladores”, Edgard Blücher, São Paulo, Brazil, 266 p.
Sullerey, R.K. and Kumar, S., 1984, “A Study of Axial Turbine Loss Models in a Streamline Curvature Computing Scheme”, ASME Journal of Engineering for Gas Turbine and Power, Vol. 106, pp. 591-597.
Ueda, T., 1982, “Improvement of Hydraulic Turbine Efficiency”, Fuji Electric Review, Vol. 28, Nr. 2, pp. 34-40.
Voith, 2004, “Brochures”, Voith Siemens Hydro Power Generation GMBH & CO. KG., Heidenheim, Germany.

11. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.