Zeitschrift: Wasser Energie Luft = Eau énergie air = Acqua energia aria

Herausgeber: Schweizerischer Wasserwirtschaftsverband

Band: 95 (2003)

Heft: 1-2

Artikel: A review on turbine design

Autor: Brekke, Hermod

DOI: https://doi.org/10.5169/seals-939431

Dank
Diese Untersuchung wurde durch die finanzielle Unterstützung der KTI (Kommission für Technologie und Innovation) sowie der Firmen Sulzer Pumpen AG, VA Tech Hydro AG und Wilo GmbH ermöglicht. Dank der guten Zusammenarbeit mit den oben genannten Firmen konnte das Projekt zielorientiert abgewickelt werden.

Literaturverzeichnis

Ref. 1 *Brennen E. Ch.,* K.-D., «Hydrodynamics of Pumps», Concepts ETI and Oxford Science Publications, 1994.
Ref. 2 *Childs, D. W.,* «Turbomachinery rotor dynamics, phenomena, modeling and analysis», J. Wiley & Sons, New York, 1993.
Ref. 3 *Geis, H.,* «Experimentelle Untersuchungen der Radseitenverluste von Hochdruck-Wasserturbinen radialer Bauart», Dissertation, TU Darmstadt, 1984.
Ref. 4 *Lauer, J.,* «Einfluss der Eintrittsbedingung und der Geometrie auf die Strömung in den Radseitenräumen von Kreiselpumpen», Dissertation, TU Darmstadt, 1995.
Ref. 5 *Möhring U. K.,* «Untersuchung des radialen Druckverlaufes und des übertragenen Drehmomentes im Radseitenraum von Kreiselpumpen bei glatter, ebener Radseitenwand und bei Anwendung von Rückenschaufeln», Dissertation, TU Carolo-Wilhelmina zu Braunschweig, 1976.
Ref. 6 *Staubli T., Bissig M.,* «Numerically calculated rotor dynamic coefficients of a pump rotor side space», Int. Symp. on Stability Control of Rotating Machinery (ISCORMA), South Lake Tahoe, August 2001.
Ref. 7 *Moore J. J., Palazzolo A. B.,* «Rotordynamic Force Prediction of Whirling Centrifugal Impeller Shroud Passages Using Computational Fluid Dynamic Techniques», International Gas Turbine and Aeroengine Congress and Exhibition, 99-GT-334, Indianapolis, 1999.
Ref. 8 *Iwatsubo, T., Sheng, B. C., Matsumoto, T.,* «An experimental study on static and dynamic characteristics of pump annular seals», NASA CP 3026, 5th Workshop on Rotordynamic Instability Problems in High-Performance Turbomachinery, Texas, A&M University, 1988, pp. 229–251.
Ref. 9 *Amoser, M.,* «Strömungsfelder und Radialkräfte in Labyrinthdichtungen hydraulischer Strömungsmaschinen», Dissertation, ETH Zurich Nr. 11150, 1995.

Anschrift der Verfasser
Prof. Dr. *Thomas Staubli, Matthias Bissig*, Hochschule für Technik und Architektur Luzern, Technikumstrasse 21, CH-6048 Horw.

A review on turbine design

Hermod Brekke

Abstract

Runners for hydraulic turbines are normally designed by experienced experts according to the company traditions. He will use the best available runner from previous model turbine tests as a base for a new possibly improved runner.
The introduction of the non-traditional X-blade runner for Francis turbines in 1996 has reduced the cross flow and improved the dynamic behaviour at part load for low-head and medium-head turbines. A discussion is given on the philosophy and theory behind this design that has not been based on traditional designed existing runners.
For Pelton turbines the introduction of a thinner and steeper inlet of the buckets and increased accuracy of model turbine production have given an increased model efficiency. However, often a lower step-up of efficiency from model to prototype is observed. A discussion on Pelton design will be given.

1. Introduction

CFD analysis is necessary in order to optimise the flow regimen in a Francis runner after the primary geometry has been decided.

If the goal is a complete new design, it is necessary to create the geometry by means of another tool than a CFD programme.

The background for this paper is to present the authors experience in using classic turbine theory in the work of creating the basic geometry of a Francis runner. The classic equations can be computerised by for example Excel programmes in order to create the geometry of crown, band and blades. Example of such runners may be illustrated by the pressure balanced so-called X-blade runner that was created by analytical calculation of the stream wise blade angles and the blade lean angle normal to the stream lines. No existing runner geometry was used to create this runner which was selected for 8 turbines for the Three Gorges project in China. The procedure to create such runners and the possibility to computerise this procedure has been the subject for this paper.

Another background for this paper is the authors experience in the improved research work on Pelton model turbines that will be presented because the model efficiency has been increased dramatically while some

Résumé

Les roues de turbines hydrauliques sont normalement conçues par des experts expérimentés selon les traditions de la compagnie. L'expert utilisera la meilleure roue disponible d'après des tests sur modèles précédents comme référence pour l'amélioration possible d'une nouvelle roue.
L'introduction de la roue non-traditionnelle "X-blade" pour les turbines de Francis en 1996 a réduit considérablement l'écoulement transversal et amélioré le comportement dynamique à charge partielle pour les turbines basse chute et moyenne chute. Une discussion est donnée sur la philosophie et la théorie derrière cette conception qui n'a pas été basée sur les roues existantes conçues traditionnellement.
Pour les turbines Pelton, l'introduction d'une entrée plus mince et plus inclinée des augets et l'augmentation de la précision de la production de modèles d'essai ont donné une augmentation du rendement du modèle. Cependant, une augmentation inférieure de l'efficacité du modèle au prototype est souvent observée. Une discussion sur la conception des turbines de Pelton sera donnée.

disappointments have been observed on prototypes when comparing model and prototype.

2. Francis runner design

Traditionally the analytical equations presented at the universities are not developed for practical use in industrial design of turbine runners. However, some practical applications may be developed for the preliminary design of for example Francis turbine runners without looking at existing runners. Some of the equations that was used for the design of the so-called X-blade runners prior to the fine tuning by CFD analysis will be shown in the following:

The well-known Euler equation written by dimensionless reduced velocities will be (See nomenclature):

$$\eta_h = (U_1 C_{U1} - U_2 C_{u2})/(gH) = 2(\underline{U}_1 \underline{C}_{u1} - \underline{U}_2 \underline{C}_{u2}) \quad (1)$$

(At the design point $C_{u2}=0$)

The equation for the outlet flow from the guide vanes towards the runner blade inlets may be expressed as follows:

The equation of continuity:

$$\underline{C}_m = \Delta \underline{Q}/(2\pi r \cos X \, \Delta l) \quad (2)$$

(note reduced values)

Here $\Delta \underline{Q}$ is the flow between two stream surfaces dividing the flow into sections with equal portion of the flow. The distance between two surfaces will be $\Delta 1$ along the outlet edge of the guide vanes.

The swirl flow variation at the vertical outlet edge of the guide vanes may be expressed by following equation:

$$\frac{d((r\underline{C}_u)^2)}{dl} = -2\underline{C}_m^2 r^2 \cos^2\alpha \cos X/\rho \quad (3)$$

(Note: α/α_g, see eq. 4)

The variation of meridian flow that is crossing the vertical outlet edge of the guide vanes must fulfil eq. 2 and eq. 4.

Because of the slope δ (=X) of the flow angle $\tan\alpha g = \tan\alpha \cos X = (C_{m0}/C_{u0}) \cos X$ then equation 4 yields:

$$\underline{C}_{m0} = \underline{C}_{u0} \tan\alpha_g / \cos X \quad (4)$$

If $\underline{C}_{m0}$ for $r = r_0$ in eq. 2 is different from C_{m0} in eq. 4, the distance between the stream surfaces must be adjusted by iteration in order to obtain an acceptable small deviation.

Here (α) is the outlet angle of the guide vanes and the value of (X) is the slope of the stream surfaces at the vertical outlet edge of the guide vanes (See Fig. 1).

The main problem during the design of an optimal high specific Francis runner is that the inflow from the guide vanes has a variation in the swirl from crown to band and also a variation of the meridian velocity. The variation of the swirl flow at the runner outlet may be reduced to some extent by the runner blades, by utilising the blade lean angle in order to make a pressure balanced runner and thus improve the flow regimen.

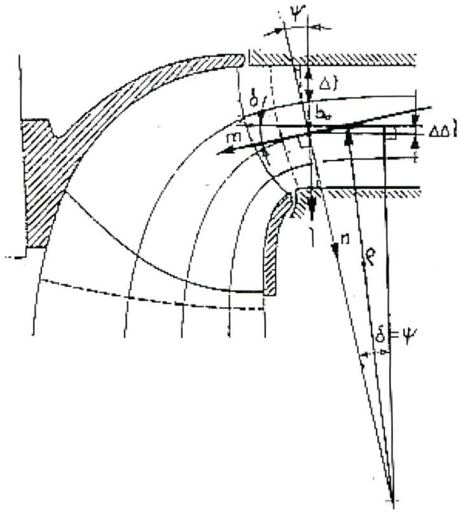

Fig. 1. Guide vane outlet-flow towards the runner inlet for calculation of the blade-inlet angles for the different stream surfaces. (Note: n-direction is normal to the meridian flow direction with positive value towards the centre of the curvature radius ρ.)

The goal for the pressure balancing work is to increase the pressure towards the band and increase the relative velocity at the crown by means of adjustment of the blade lean angle and thus reduce the cross flow on the pressure side of the blades.

A runner with reduced cross flow at best efficiency will normally also have less dynamic problems at part load because the stagnation and reversed flow will be reduced and thus giving less impact on the flow regimen.

During the design of a runner, an optimal blade lean angle may be calculated by means of three governing equations valid for potential flow i.e. infinite number of blades. These equations have been presented before (Ref. 1), but they will be repeated in this paper as an illustration of the procedure of the design:

The equilibrium of forces, Newton's second law, at the regarded point for calculation based in the meridian velocity:

$$\frac{d\underline{h}}{dn} = -\, f(R, r, \beta, \delta, \underline{C}_m, \varpi)\tan\Theta + f(r, \rho, \beta, \delta, \underline{C}_m, \varpi) \quad (5)$$

(See Ref. 1)

The Rothalpy equation from the blade inlet to the regarded point for calculation yields (Used for comparison with the hydraulic pressure variation in eq. 5):

Nomenclature

Term	Symbol	Definition
Net head	H	m
Reduced absolute velocity	$\underline{C}$	$= C/(2gH)^{0.5}$
Meridian absolute velocity component	C_m	m/s
Tangential absolute velocity component	C_u	m/s
Speed	n	rpm
Angular velocity	ϖ	rad/sec
Reduced angular velocity	$\underline{\omega}$	$= \varpi/(2gH)^{0.5}$
Reduced flow	$\underline{Q}$	$= Q/(2gH)^{0.5}$
Speed number	$\underline{\varsigma}$	$= \underline{w}\sqrt{\underline{Q}}$
Circumference velocity $= \varpi r$	U	m/s
Reduced circumference velocity	$\underline{U}$	$= U/(2gH)^{0.5}$
Outlet angle of guide vanes (Eq. 4)	α_g	rad.
Runner blade angle (Relative flow angle)	β	rad.
Absolute flow angle (Eq. 3, Eq. 4)	α	rad.
Stream surface slope angle	χ	rad.
Slope angle of stream surface in runner	δ	rad.
Radius of regarded point	r	m
Radius of blade curvature	R	m
Blade lean angle	Θ	rad.
Angle of meridian section	ϕ	rad.
Reduced hydraulic pressure	$\underline{h}$	$= h/H$
Angle between radial plane and particle	θ	rad.
Meridian curvature radius of stream surface	ρ	m
Denotations:	0	
Outlet of guide vanes	1	
Inlet of runner blades		
Outlet of runner blades	2	

$$h = \varpi^2 r^2 - C_m^2/\sin^2\beta + (1 - 2U_1C_{u1}) - J \qquad (6)$$

($0 \leq J \leq 0.02$ is the loss from blade inlet to the regarded point)

The equation of continuity to determine the meridian velocity component for numerical iteration was given also in eq. 2 where the distance between crown and band at the guide vane outlet was divided in stream surfaces. However, inside the runner the thickness of the blades (t) must be taken into consideration and correction for the blades displacement must be made as shown in eq. 7.

In eq. 7 (n) is the direction normal to the meridian direction (m), and (dn) is the distance between two neighbouring stream surfaces and N is the number of "stream channels" between the stream surfaces. In each stream channel the flow is the same i.e. $\underline{Q}/N = \Delta\underline{Q}$. The equation of continuity yields:

$$\underline{C} = (\underline{Q}/N)/(2\pi r\ \Phi dn) \qquad (7)$$

(Where $\Phi = (1 - (tN/2\pi r)\,(1 + 1/\tan^2\beta)^{0.5}$, and r = radius to the regarded point.)

Equations 5, 6 and 7 may be solved by starting with calculating ($\underline{h}$) by means of eq. 6 for the middle stream channel and calculating ($\underline{h} + (d\underline{h}/dn)\Delta\underline{n}$) by means of eq. 5 for all stream channels. Then ($\underline{h}$) is calculated for all stream channels by means of eq. 6 for comparison. Then the distance (dn) must be adjusted giving new values of ($\underline{C}_m$) by iteration in a computer programme using an approach by means of elliptical shape of the meridian sections of the stream surfaces.

The iteration process will be simplified by letting ($\underline{h}$ = constant) in eq. 6 and then letting (($d\underline{h}/dn$) = 0) in eq. 5.

Before the blade shape is proposed the inlet angle must been calculated as shown. In addition the outlet angle must be determined at the beginning of the process. The value of the outlet angle is based on $NPSH_A$, the rated flow and the net head that is the base for the outlet diameter and the outlet angle at the band and the variation of the angle from band to crown. A detailed description of this process will not be included in this paper.

The optimum value of ($d\underline{h}/dn$) should in general be zero for potential flow, i.e. infinite number of pressure balanced blades. The optimum value of ($d\underline{h}/dn$) is depending on the geometry of crown and band that must be chosen before the blade shape is determined.

The values of (Θ) and (β) can be calculated from the values ('Θ)and ('β) as shown in Fig. 2 or in contrary ('Θ) and ('β) can be calculated from calculated values of (Θ) and (β). The optimum values of (Θ) can be calculated from eq. 5 for a chosen optimum value of ($d\underline{h}/dn$) that may be ($d\underline{h}/dn$) = 0 or another chosen positive value in order to increase the pressure at the band.

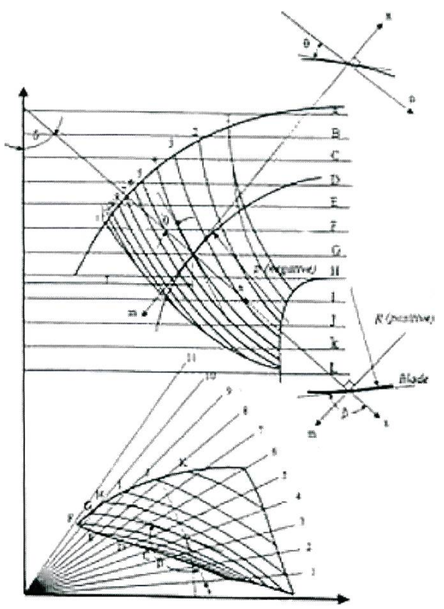

Fig. 2. The blade shape and geometry for the runner design. (The illustrated runner is not a pressure balanced runner.)

Further the value of (β) along the blade may be expressed by a third order function of $X = r\Phi$ along the circumference as shown in eq. 8.

$$m = aX + bX^2 + cX^3 \qquad (8)$$
where $X = r\ \Phi$

(Φ = the angle of the meridian sections measured from the blade outlet to the regarded point on the stream surface for the calculated value of $d\underline{h}/dn$.)

3. Calculated results

In Fig. 3a is shown a manually calculated and drawn pressure balanced Francis runner blade for a medium specific speed turbine.

The inlet and outlet angles of the presented blade have not had the final adjustment of inlet- and outlet angles, but will serve as input for making the grid for the CFD analysis. The main shape of the blade represents a typical pressure balanced blade created by the presented theory.

In Fig. 3b is shown the result of a preliminary programme for automatic drawing of a Francis turbine blade. Such automatic drawing system may be linked to the presented theory in this paper forming a complete design programme.

4. Pelton turbine design

The Pelton runner design has traditionally been based on experiments based on a simplified theoretical analysis only.

However, a graphical time consuming analysis has been used in Norway at Kvaerner since 35 years ago.

At the Norwegian University of Science and Technology as well as in other universities and the industry, research work is going on the numerical analysis of unsteady free surface flow by means of CFD.

So far successful results has been obtained in CFD modelling of the velocity distribution in the jet compared with pitot-measurements and Laser velocity-meter measurements. For the non-stationary free surface flow in the buckets there is still more work to be done, but promising results has been obtained. The main problem is the movement of the jet passing through a grid system fixed to the Pelton bucket.

The traditional graphical analyses made 35 years ago, also gave a good agreement with the experimental stroboscopic studies and high speed photos. The basic theory for the graphical analysis was built on

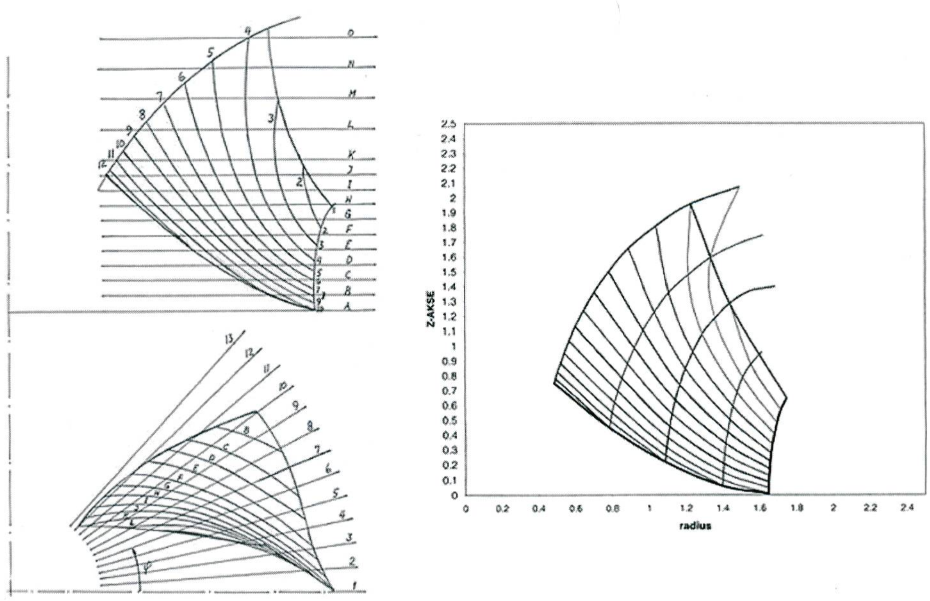

Fig. 3. Left, a manually calculated and drawn Francis X-blade. Right, a preliminary automatic drawing of a Francis runner.

the fact that the resultant acceleration must be normal to the water surface of the water in the bucket.

The governing equations for this analysis are (Ref. 2):

Acceleration in radial direction (X) of the bucket:

$$a_x = \frac{d^2X}{dt^2} + \varpi^2 R \cos\theta + 2\varpi\, C_y \quad (9)$$

Acceleration normal to the buckets outlet plane (Y):

$$a_y = \frac{d^2Y}{dt^2} + \varpi^2 R \sin\theta + 2\varpi\, C_x \quad (10)$$

Acceleration in axial direction (Z):

$$a_z = \frac{d^2Z}{dt^2} \quad (11)$$

In Fig. 4 a stroboscopic photo with exposure time $1.2 \cdot 10^{-6}$ sec. is illustrating the problem caused by water leaving through the buckets inlet. Such photos were used for verification of the graphical analysis based on eq. 9–eq. 11 (Ref. 2).

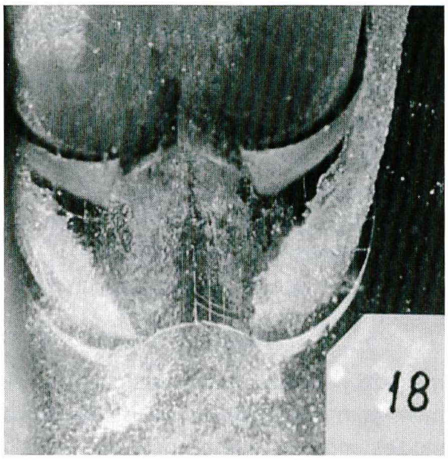

Fig. 4. Stroboscope photo taken in 1967 (exposure time $1.2 \cdot 10^{-6}$ sec.) (Courtesy: Kvaerner Brug A.S.) (Ref. 2).

However, on the experimental side improvements has been made by more accurate bucket reproduction and better tools to define and detect the effect of small changes in the geometry. Also high-speed video and measurements of the pressure distribution on the buckets' inside surface, have given important information to the designers.

This work has given an improvement of the model efficiency that will exceed 92% for a low specific speed runner.

However, very few prototypes except for some turbines with low specific speed runners have proven an efficiency exceeding 92%.

It should also be mentioned that lifting up the buckets' entrance lips and the splitter has increased the efficiency by reducing the outlet flow through the buckets inlets, but then the margin against droplet erosion and cavitation pitting on the back-side of the buckets has decreased. To compensate for the reduced margin against droplet erosion and cavitation pitting, the entrance lips have been made as thin as possible. The reduced thickness has also given a contribution to the efficiency.

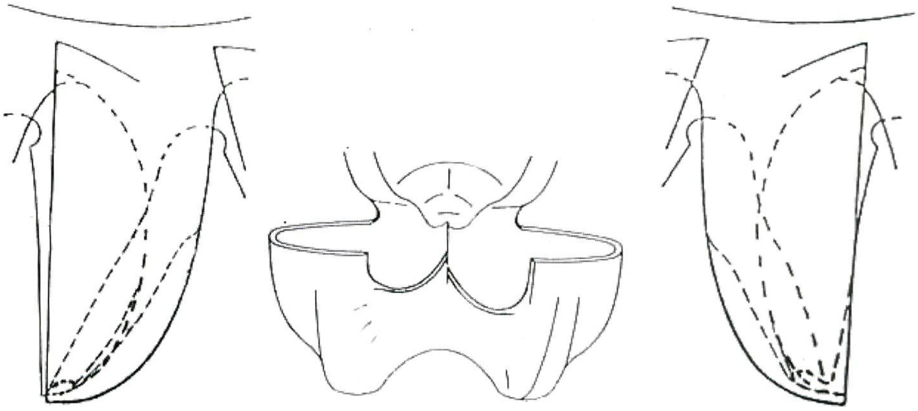

Fig. 5. Illustration of the improved inlet of a modern Pelton bucket, left side, and illustration of an "old" traditional design, right side.

However, the thickness reduction will be limited by the structural strength and the wear caused by sand-erosion. The improved design of the Pelton-bucket inlet in order to obtain higher efficiency, is illustrated in Fig. 5.

5. Conclusion

CFD is a needed tool for analysis of the flow in a Francis runner and in some cases model tests will not be necessary to prove the ability of the runner. However, before the CFD analysis the runner geometry is needed. In the majority of cases the experienced manufacturers will start the CFD analysis on a slight modification of an existing successful runner that have proved a performance close to the requirement for the new runner.

In the paper a procedure is shown for creating the geometry for a new runner based on classic mathematical formulas for potential flow. By means of computers a geometry programme may be established creating a pressure-balanced runner that may have the geometry different from the typical "company runners". An example of such runner design is the so-called X-blade runners that have been chosen for 8 turbines in Three Gorges Power Plant in China.

For the theoretical work on Pelton turbines progress is made on CFD analysis of the free surface jet, and work has been started on the free surface none stationary flow in the buckets. However, so far the only theoretical analysis that has been completed is the graphical analysis backed up by stroboscopic or high-speed video model studies.

Nevertheless by lifting up the jet entrance portion and make the inlet thinner the model efficiency have exceeded 92% at present, but very few prototypes have proven the same progress. In the future the main task will be to solve the none stationary free surface flow by CFD analysis, but at the time being a suitable grid system for the none stationary problem has not been found to the authors knowledge.

References

Ref. 1 – *H. Brekke* "Francis Runner Design for Optimum Cavitation Performance" Hydropower 96. Oct. 28.–Nov. 2, Beijing.

Ref. 2 – *H. Brekke* "A General Study on the Design of Vertical Pelton Turbines". 35 years of Turboinstitute. Ljubljana 84.

Anschrift des Verfassers

Prof. *Hermod Brekke*, the Norwegian University of Sciene and Technology, N-7026 Trondheim, Norway.

Dieser Beitrag ist ein Nachdruck aus den Proceedings zum XXI. IAHR Symposium on Hydraulic Machinery and Systems, welches vom 9. bis 12. September 2002 in Lausanne durchgeführt wurde.