Kaplan Turbine Performance Prediction Using CFD: an Artificial Neural Network Approach

A. Arnone, M. Marconcini, F. Rubechini, A. Schneider
“Sergio Stecco” Dept. of Energy Engineering
University of Florence
via di Santa Marta, 3, 50139, Firenze, Italy
andrea.arnone@unifi.it

G. Alba
ZECO S.r.l.
via Astico, 52C, 36030, Fara Vicentino (VI), Italy
giovanni.alba@zeco.it

Introduction

In this work a response surface approach was used for predicting the hill chart of a small horizontal shaft Kaplan turbine. To this aim, artificial neural networks (ANN) were chosen as a fast, reliable, and computationally inexpensive tool. The training of the ANN was based on computational fluid dynamics (CFD).

The optimization of the runner-guide vane stagger correlation is a time consuming and expensive task when obtained either on experimental models or directly on the power plant. The use of CFD coupled with ANN may represent an attractive alternative. In order to obtain a more general prediction tool it was decided to characterize the turbine performance coupled with a generic draft tube. The draft tube is a key component for the performance of small head power plants. As a matter of fact, the kinetic energy recovery may represent a considerable fraction of the total head, thus strongly affecting the efficiency. It presents a very complex flow environment characterized by unsteady, large scale vortices, and the prediction of its performance is a challenging task for the CFD. Moreover, in the refurbishment and repowering of hydro power plants, the turbine may often be coupled with a pre-existing draft tube. For these reasons the draft tube was not included in the computational domain, and a simple 1D model was used to account for its influence on the turbine.

The three-dimensional version of the TRAF code developed at the University of Florence, coupled with a two equation turbulence closure has been used to predict the turbine flow features and the operating characteristics. The code exploits the artificial compressibility concept to work with incompressible flows. Steady multirow viscous single-phase analysis has been applied to compute the flow field from the inlet struts to the runner exit. The CFD results were used for the training of a feed-forward artificial neural network with two hidden layers. As far as the training is concerned, a gradient based back propagation method was employed. In order to improve the generalization ability, a hybrid network made by multiple trained neural networks was used. The considered input parameters were the stagger angles of the runner and the guide vane, the total head and the draft tube recovery coefficients. The outputs of the neural network were the computed mass flow rate and the hydro power plant efficiency.

The proposed procedure was applied to an existing power plant in order to optimize the runner-guide vane stagger correlation for all runner positions. Comparisons with the measurements obtained on the hydro power plant are presented and discussed. The predicted coupled positions of the guide vane and runner blades were successfully verified all over the range of the operating mass flows.

1. Computational Environment

The multi-row, multi-block release of the HYDROMS code (Arnone et al., 1994), which is the incompressible single-phase version of the TRAFMS code (Arnone, 1994, 1995), a 3D solver originally developed at the University of Florence for compressible turbomachinery flows, was used in the present work. The concept of artificial compressibility of Chorin (1967) is used to handle incompressible flows by a time-marching approach. The time derivative of pressure, weighted by an artificial compressibility coefficient, is added to the continuity equation in order to obtain an unsteady formulation for the three-dimensional Reynolds Averaged Navier-Stokes (RANS) equations written in conservative form in a curvilinear, body-fitted coordinate system.

1.1 Numerical Scheme

The spatial discretization of the equations is based on a finite volume scheme cell-centered scheme. A blended second- and fourth-order artificial dissipation model (Jameson, Schmidt and Turkel, 1981), together with an eigenvalue scaling (Martinelli and Jameson, 1988) is used in order to minimize the amount of artificial diffusion inside the shear layers. The equations are advanced in time using an explicit four-stage Runge-Kutta scheme, until the steady state solution is reached. In order to reduce the computational cost and speed up convergence to the steady solution, four computational techniques are employed: local time-stepping, implicit residual
smoothing, multigrid full approximation storage (FAS) and grid refinement.

Inflow and outflow boundaries are treated according to the theory of characteristics: the flow angles and the total pressure are imposed at the first row inlet, while the static pressure is determined by a first-order extrapolation from the interior points. At the last row outlet, static pressure is prescribed and the velocity components are extrapolated from the interior points. The link between rows is handled by means of mixing-planes. Consecutive rows have a common interface plane and the match is provided through appropriate calculation of phantom cell values, keeping the span-wise distribution while averaging in the blade-to-blade direction.

The two equation $k-\omega$ model of Wilcox (1988, 2008) was used for turbulence closure. The freestream value of the specific turbulence dissipation rate $\omega_\infty$ is determined by using the following estimate proposed by Menter (1992):

$$\omega_\infty = 10 \frac{V_\infty}{L},$$

being $V_\infty$ the inlet flow velocity and $L$ the computational domain length. The freestream eddy viscosity ratio is set to $\nu_\infty/\nu = 50$, and the turbulent kinetic energy is then calculated from the definition of the eddy viscosity $k_\infty = \nu_\infty \omega_\infty$. At the solid walls, the value of $k$ is set to zero and the value of $\omega$ is computed using Wilcox’s roughness model for smooth surfaces (Wilcox, 2008).

1.2 Artificial Neural Networks

The determination of the optimum stagger angle law in a Kaplan turbine is an optimization problem with two degrees of freedom. The direct calculation of all the possible combinations leads however to a quite large number of calculations. For example, choosing an angular spacing of 5° in a range from 10° to 80° for the guide vane, and a resolution of 2° for the runner, may require up to 400 CFD calculations (e.g. Gehrer et al., 2004, Benigni et al., 2006). An attractive alternative is represented by a response surface approach, which allows a reduced sampling of the operating range. The strategy is often referred to as meta-model technique since it provides a “model of the model” (Kleijnen, 1987). To this aim, artificial neural networks (ANN) were chosen as a well-known approach to fit a wide class of objective functions. Sobol’s sequence was used to generate the training dataset. With respect to a random sampling, the Sobol sequence provides a more effective coverage of the design space under investigation. In order to improve the generalization capability, a hybrid network made by multiple trained neural networks was used (e.g. Rai, 2002, Rubechini et al., 2009).

The considered input parameters are the stagger angles of runner and guide vane, the total head and the draft tube recovery coefficients. The outputs of the neural network are the mass flow rate and the efficiency. The stagger angles only are processed as optimization variables, while the other input parameters are variable boundary conditions. As far as the output parameters are concerned, the mass flow is a constraint while the efficiency is the objective function.

2. Application and Discussion

2.1 Experimental Analysis

The experimental campaign was carried out on the real power plant according to the CEI EN 60041 procedure (see references). Tests were performed for five different mass flows, ranging from 40% to 100% of the nominal flow rate. For each flow rate the stagger angles were optimized during the tests. Starting from the CFD-ANN predicted values the stagger angles were varied to check if an additional refinement was achievable.

Static pressure was measured in section 1, upstream to the guide vane. The discharge basin level was measured at section 3 with an ultrasonic transducer (Endress+Hauser). The volume flow was measured with an ultrasonic transducer with four paths (Rittmeyer). The group efficiency was then obtained from the measured electric power $P_e$:

$$\eta_g = \frac{P_e}{\rho g \bar{Q} H_n}$$

where $H_n$ is the net total head. Each measured quantity was acquired with a sampling of 5s on a period of 15min, the averaged values were then used for the efficiency definition and the rms values were monitored. The volume flow was measured with a sampling of 30s over the same period. An example of the time evolution of the measured quantities is given in Fig. 1(a)-(b). The total measurement uncertainty on the efficiency is estimated about ±1.5%.

2.2 Computed Results

The horizontal-shaft “S-shaped” tubular turbine under investigation includes 4 struts, 16 guide vanes and 5 runner blades. The runner specific speed is $n_s=612$ near the design point, where:
Fig. 1. Time evolution of the measured quantities (a) volume flow $Q$ (b) electric power $P_e$.

Fig. 2. (a) Turbine meridional plane ($x$-$r$) (b) schematic of the runner tip clearance mesh (c) three-dimensional view of the computational grid.

$$n_r = n \frac{P}{H^{5/4}}$$

with $n$ [rpm], $P$ [kW] and $H$ [m]. The draft tube, not included in the computational domain, has mainly an axial development with a transition from a circular section at the runner exit to a rectangular section at the diffuser exit.

A meridional section of the computational domain is shown in Fig. 2(a). Each component was discretized using an H-type single-block non-periodic elliptic grid using an in-house developed grid generator. For the case of a Kaplan turbine, the leading aspects of the blade geometry are the twist and the stagger. At low flow rate operating condition the stagger angle become very high and the removal of mesh periodicity allows to minimize the mesh skewness inside the blade passage. A 141x65x81 grid was used for the stay vanes in the stream-wise, pitch-wise and span-wise directions respectively. The guide vane geometry was modelled including the casted thickening of the hub with a 173x61x81 grid. The runner grid has 225x69x81 points. The tip clearance between the runner blade and the shroud was discretized with 8 grid cells, the clearance region was handled by pinching the blade (e.g. Storer and Cumpsty, 1991, Marconcin et al., 2008) and the tip gap is fully gridded (see Fig.2(b)). The value of the $y^+$ for the first grid point above the wall was between 1.0 and 4.0 for all the blades. A three-dimensional view of the computational mesh at the nominal blade stagger angle, for a total of about 2.85 million points, is shown in Fig. 2(c). The CPU time required for each computation was about 3 hours on a CPU Xeon 3.6GHz.

A total of 50 CFD runs were performed to build the training dataset. Ten neural networks with different architecture, training subset, and weight vector initializations, were used for an effective hybridization. A simple feed-forward topology with up to 6+4 neurons in the two hidden layers was chosen. A gradient-based back-propagation method was used for the training. The generalization ability of the hybrid network was evaluated by computing the prediction error over an independent validation set of 5 CFD runs. An average error of 0.07% on
the prediction of the efficiency and of 0.0012% on the prediction of the mass flow rate was found.

The turbine efficiency is computed using the following definition:

$$\eta_t = \frac{P_t}{\gamma Q(H_1 - H_3)}$$  \hspace{1cm} (3)

The draft tube is modelled through the kinetic energy recovery coefficients $c_{pm}$ and $c_{p\theta}$ for the meridional and the swirl velocity respectively computed at the runner exit section 2, the static pressure at the runner exit $p_2$ was automatically changed during the computations in order to match the available power plant total head $H_1 - H_3$:

$$H_3 = \frac{1}{2g}\left(c_{pm}c_{2m}^2 + c_{p\theta}c_{2\theta}^2\right) + \frac{p_2}{\gamma}$$  \hspace{1cm} (4)

The recovery coefficients were allowed to vary in the range $c_{pm}=0.60-0.90$ and $c_{p\theta}=0.0-0.4$ respectively. The turbine power is obtained from:

$$P_t = \int_S \left(p + \tau_w\right)r\omega \cdot dS_{\theta}$$  \hspace{1cm} (5)

where $p$ and $\tau_w$ are the static pressure and the wall shear stress respectively, $dS_{\theta}$ is the surface element normal to the circumferential direction, $\omega=2\pi n/60$ is the angular speed, and the integration is extended to all the rotating surfaces.

Figure 3 shows the computed flowfield for the nominal operating point $Q/Q_{max}=100\%$. The computed static pressure contours on the solid surfaces are depicted in Fig.3(a), the absolute velocity contours on a blade-to-blade surface near mid-span are shown in Fig. 3(b).

The application of the proposed methodology to the power plant described previously required choosing the draft tube recovery coefficients. On the basis of correlations for diffusers (Idel’cik, 1986) and the computed flow distortion at the runner exit, the recovery coefficients were estimated to $c_{pm}=0.7$ and $c_{p\theta}=0.3$.

The predicted performance are compared to the measured ones in Fig. 4. As far as the turbine efficiency is concerned, Fig. 4(a), the envelope of the predicted curves is represented as a dash line. A fairly good agreement is found in the range 70%-100%, at lower mass flows the discrepancy is around 1%. Figure 4(b) compares the optimum stagger angle law obtained on the power plant versus the CFD-ANN prediction. Here the maximum deviation over the investigated range is less than 2°.

![Fig. 3. Computed flow field $Q/Q_{max}=100\%$: (a) static pressure contours, (b) absolute velocity contours on a blade-to-blade plane near mid-span.](image)
3. Concluding Remarks

A three-dimensional Navier-Stokes solver for incompressible flows was used to investigate a small horizontal-shaft Kaplan turbine. The main goal was to optimize the runner-guide vane stagger correlation and thus the power plant performance on the operating range. The proposed strategy couples CFD analyses and artificial neural networks (ANN) as a powerful approach which allows us to minimize the computational effort and reduces the need for time-consuming and expensive experimental tests. The CFD-ANN methodology was applied to an existing power plant and the results were compared to the measurements. The predicted stagger angles were verified on the power plant and an additional refinement of the stagger correlation was performed during the tests in order to check if a deviation from the computations occurred. A good agreement between predicted and measured stagger law was found over the investigated flow rate range, with a maximum discrepancy around 2°.

While this work has shown that the proposed methodology potentially offers an economical and accurate mean of providing affordable engineering predictions of optimum runner-guide vane stagger correlation, the validation arguably needs to cover a broader range of applications to arrive at secure conclusions about the reliability of the CFD-ANN approach.

References

5. CEI EN 60041, “Prove di collaudo in sito per la determinazione delle prestazioni idrauliche delle turbine idrauliche, delle pompe di accumulazione e delle pompe-turbine”.


The Authors

Andrea Arnone is Full Professor at the “Sergio Stecco” Dept. of Energy Engineering at University of Florence, where he earned his undergraduate degree in Mechanical Engineering. After completing his Ph.D. at the University of Bologna, he worked as an associate researcher at the Institute for Computational Mechanics in Propulsion (ICOMP), NASA Lewis Research Center. Since 1990, he has been a member of the Faculty of Mechanical Engineering at the University of Florence. His research involves the use of CFD in turbomachinery design and analysis. He has developed 2D/3D codes that are currently used by several industries and research centers.

Michele Marconcini completed his undergraduate degree in Mechanical Engineering at the University of Florence in 1996. He received his Ph.D. in Energy Engineering from the same University in 2000. From 2007 he is Assistant Professor at the “Sergio Stecco” Dept. of Energy Engineering. His research activity mainly concerns the application of CFD techniques to advanced aerodynamic design of aeronautical and industrial turbomachines.

Filippo Rubechini completed his undergraduate degree in Mechanical Engineering at the University of Florence in 2000. In 2004, he received his Ph.D. in Energy Engineering from the same University. From 2000 he works as Research Fellow at the “Sergio Stecco” Dept. of Energy Engineering. His current activity mainly concerns the use of CFD techniques for the design of industrial turbomachinery components.

Andrea Schneider completed his undergraduate degree in Civil Engineering at the University of Florence in 2000. From 2000 he works as Research Fellow at the “Sergio Stecco” Dept. of Energy Engineering. His current activity mainly concerns the application of optimization techniques to industrial turbomachines and racing yacht aerodynamics.

Giovanni Alba completed his degree in Mechanical Engineering at the University of Padova in 2001. After a three-year-background in molding industry for automotive; from 2004 he is Mechanical engineering manager in ZECO S.r.l. His current activity mainly concerns the design of hydroturbines, mainly about the fluid-mechanics aspects; feasibility study, design and check of the experimental set-up, data acquisition and analysis.