Wicket gate trailing-edge blowing: A method for improving off-design hydroturbine performance by adjusting the runner inlet swirl angle

B J Lewis, J M Cimbala, and A M Wouden
Department of Mechanical and Nuclear Engineering, The Pennsylvania State University, University Park, PA, USA
E-mail: bj15176@psu.edu, jmcimbala@psu.edu

Abstract. At their best efficiency point (BEP), hydroturbines operate at very high efficiency. However, with the ever-increasing penetration of alternative electricity generation, it has become common to operate hydroturbines at off-design conditions in order to maintain stability in the electric power grid. This paper demonstrates a method for improving hydroturbine performance during off-design operation by injecting water through slots at the trailing edges of the wicket gates. The injected water causes a change in bulk flow direction at the inlet of the runner. This change in flow angle from the wicket gate trailing-edge jets provides the capability of independently varying the flow rate and swirl angle through the runner, which in current designs are both determined by the wicket gate opening angle. When properly tuned, altering the flow angle results in a significant improvement in turbine efficiency during off-design operation.

1. Introduction
As water is a precious resource in most global locations, it is essential that hydropower technology utilize the available water with the highest possible efficiency. The present work presents a concept for improving the performance of traditional hydropower generation by the addition of water jets to the trailing edge of the hydroturbine wicket gates. With the ever-increasing penetration of alternative electricity generation, it will become even more common in the future to operate hydroturbines under off-design conditions to maintain stability in the electric power grid. Therefore, particular emphasis must be given to improving the off-design performance.

This paper demonstrates a method for improving hydroturbine performance during off-design operation by injecting water through slots at the trailing edges of the wicket gates. The authors, in [1], found that this revolutionary concept has the capability to reduce the intensity of the rotor-stator interactions by compensating for the momentum deficit of the wicket gate wakes. Of greater significance however, is the fact that wicket gate trailing-edge blowing may additionally incorporate the concept of circulation control hydrofoils, which leads to a change in the inlet flow angle to the runner. This change in flow angle provides the capability of independently varying the flow rate and swirl angle through the runner, which in current designs are both determined by the wicket gate opening angle. Trailing-edge blowing from either the guide vanes or rotor blades has been successfully applied to various axial flow turbofans, compressors, and gas turbines; however, no literature has been found for the application of trailing-edge blowing to centrifugal or mixed-flow devices, such as Francis hydroturbines.
2. Previous Work in Wicket Gate Wake Control

Of interest to the present work are the methods that have been studied to control guide vane wake dynamics and vortex shedding. The most successful methods have been to modify the shape of the guide vane trailing edge or to inject fluid from the trailing edge of the guide vanes.

2.1. The effect of trailing edge shape

The shape of the guide vanes is an important contributor to the intensity of the rotor-stator interactions [2]. In 1959, Donaldson [3] compared the flow induced vibration frequency and amplitude of 10 different trailing edge shapes (see Figure 1). Donaldson’s study concluded that compared to the blunt trailing edge shape each of the trailing edge shapes reduced the vibration amplitude, except for the rounded and pointed designs (5, 6, and 7). The sharp beveled designs (8 and 9) and the Donaldson design (10) produced vibrations that were below the sensitivity of the measurement equipment. Other studies by Blake et al. [4] measured the amplitude for these shapes to be between 0.3% and 1.5% of the blunt trailing edge.

![Figure 1. Vibration amplitude of various hydrofoil trailing edge shapes, normalized by the vibration observed for a blunt trailing edge. The sharp beveled designs (8, 9) and Donaldson design (10) produced the minimum vibration amplitude. Source: Donaldson [3].](image)

2.2. Trailing-edge blowing

In the early 1960’s, Naudascher [5], and his students Ridjanovic [6] and Wang [7], began studying the turbulent behavior of flow in the wake of a body with hydrodynamic self-propulsion. The basic concept of self-propulsion is that the drag on the disturbing body is balanced by a source of added momentum, such as a flapping fin, propeller, or propulsion jet. When the added momentum and drag on the body are in balance, a momentumless wake is formed.

Following Naudascher’s work, significant experimental and computational research has been conducted on trailing edge blowing and momentumless wakes. Sirviente and Patel [8, 9] analyzed the wake of a slender self-propelled body, focusing on the the mixing of the shear layers. Sirviente and Patel [9] also investigated the effect of a swirling jet and showed that swirl increased the momentumless wake dissipation rate. Brucker and Sarkar [10] studied slender self-propelled bodies, adding the complexity of a stratified media. Also, the effect of swirl in the mean flow was analyzed by Chernykh et al. [11]. Cimbala and Park [12, 13, 14] investigated the turbulent structure of 2-D momentumless wakes, and found that the decay rate for the centerline velocity deficit was much faster for the 2-D momentumless wake than for the pure wake. They also found that a jet configuration using two thin slots, equally spaced from the centerline of the body, further increased that wake decay rate.
3. Design of the Wicket-Gate Trailing Edge Jet
Several designs were considered for adding water jets to the wicket gates. The original concept was to add two parallel slot jets to the existing blunt trailing edge of the vanes (see Figure 2). Dual slots were chosen based on the result of Park and Cimbala [13]. The narrow slots traversed the complete span of the wicket gate and were connected to a plenum located near the pivot axis. The location for the plenum was chosen to enable water to be supplied to the jets through the center of the wicket gate pivot, without interrupting the pivoting function of the wicket gate.

![Figure 2. Geometry for the parallel slot jets and supply channels added to the original blunt trailing edge wicket gate of the GAMM Francis Turbine. The slots traversed the complete span of the guide vane and were connected by narrow channels to a plenum located near the pivot axis.](image)

After various design iterations, a beveled trailing edge was added to the wicket gate, with dual slot jets inwardly inclined from the beveled surface (see Figure 3). The beveled trailing edge was adapted from the results of Donaldson [3], and was found to eliminate many of the unsteady features of the wicket gate wake.

![Figure 3. Final design of the trailing edge geometry for the jet channels added to the wicket gates of the GAMM Francis Turbine. The blunt trailing edge was replaced with a bevelled shape, and the jets were inclined toward the center of the vane to improve the maximum turning of the water.](image)

The optimal location and inclination angle of the jets were determined by optimizing various design parameters through a Taguchi method [15]. The Taguchi method was used in order to minimize the number of simulations required to optimize the design, by comparing each simulation through a level averaging technique. Each jet design was compared based on the effectiveness of the trailing-edge blowing in changing the swirl angle observed by the runner.

4. Numerical Methods
The unsteady flow through the hydroturbine was simulated by solving the Reynolds averaged Navier-Stokes equations, with both $k-\varepsilon$ and $k-\omega$-SST eddy viscosity turbulence models. The governing equations and boundary conditions were consistent with the industry standards. The flow field was simulated using OpenFOAM®, an open source CFD solver, which enabled the development of customized numerical solvers and pre/post-processing tools for the present simulations.
To analysis the behavior and capability of the wicket gate trailing-edge injection on the flow entering the runner, various periodic simulations of the inlet distributor channel were conducted (see Figure 4). The distributor was modeled using both a quasi-2-D domain (Figure 4(a)) and a 3-D domain (Figure 4(b)). The quasi-2-D domain was used to conduct design iterations of the trailing-edge jets, and to study the unsteady wake dynamics. The 3-D distributor passage was used to analyze the effects of wicket gate trailing-edge blowing in a realistic turbine flow passage, and to estimate the change in swirl angle distribution at the inlet of the runner due to injection. The domains extended through the runner region in order to place the outlet location a sufficient distance downstream so as to not affect the solution near the vanes. The computational domains also included the supply channels for the jets, as the authors found it was necessary to model the jet supply channel in order to correctly model the downstream wake-jet behavior [16].

![Figure 4](image.png)

(a) Quasi-2-D domain  
(b) 3-D domain

**Figure 4.** Periodic computational domains for the GAMM distributor vane channel used to analysis the effect of wicket gate injection on the downstream swirl angle. The gray region in (a) identifies the swept location of the runner blade leading edge. The Runner Inlet Axis in (b) is shown only for illustration and identifies the location of circumferentially averaged data sampling.

The distributor simulations provided an understanding of the change in swirl angle distribution at the runner inlet. The effects of the change in swirl angle on the runner performance were evaluated separately, using a periodic runner passage simulation. For simplicity, an axisymmetric diffuser was used in place of the draft tube (see Figure 5(a)). The axisymmetric diffuser enabled the flow to be assumed rotationally periodic. The effect of the blade rotation was modeled by including source terms in the governing equations for the centripetal and Coriolis body forces. These source terms were applied only to cells contained within the rotating runner region (colored red in the figure). Time-averaged velocity profiles were measured in the experiments at the runner inlet axis (see [17] and [18] for more details). These profiles were extrapolated upstream using conservation of mass and angular momentum to a radial location of 239.3 mm, and used as the inlet condition for the present simulations. This same approach was used by Nilsson and Davidson [19, 20].

The various periodic simulations were able to produce significant insights into the design and turning behavior of the wicket gate trailing-edge jets and the effect of the turning on the runner performance. However, these periodic simulations were incapable of capturing either the unsteady rotor-stator interactions (RSI), or the effect of the blowing on the unsteady performance of the turbine. Therefore, it was necessary to conduct full-wheel unsteady simulations of the complete GAMM Francis Turbine (see Figure 5(b)).
Figure 5. Interactive 3-D models of the GAMM Francis Turbine, showing the computational domains used to analyze the turbine performance. Red surfaces identify the rotating runner, and blue surfaces are stationary. Only one blade passage was included in the simulations. The complete runner wheel is shown in (a), though only one runner blade passage was included in the simulations. Note: Adobe Reader 10, or later, required to view the 3-D models.

The full-wheel turbine simulations were conducted with a grid containing approximately 65 million cells, and included the dynamic rotation of the runner wheel mesh. Sliding grid interfaces were used to couple the rotating and stationary meshes. The spiral casing was not included in the simulations, as the geometry was not available. A time-averaged inlet profile was measured in the experiments at the inlet to the stay vanes, and was applied in the simulations as a circumferentially uniform inlet profile. Though the GAMM Francis Turbine has been studied in multiple workshops, symposiums, and papers, to the authors’ knowledge, the present work is the first to model the unsteady flow through the complete turbine.

5. Results
Results are presented below for the quasi-2-D distributor passage, the 3-D periodic distributor passage, the single runner blade passage, and the complete turbine. Steady-state simulations were conducted for the 3-D distributor and runner blade passages. Unsteady simulations were conducted for the quasi-2-D distributor passage and the complete turbine.

5.1. Quasi-2D Distributor Passage
By injecting water from the wicket gate blunt trailing edge, or adding a beveled shape to the trailing edge of the wicket gate, the unsteadiness in the wake due to large von Kármán vortex shedding was significantly reduced. As shown in Figure 6, both the blunt trailing edge with blowing and the beveled trailing edge effectively eliminate the von Kármán vortices downstream.

As a quantitative comparison, the unsteady moment on the wicket gate was measured in each simulation (see Figure 7). The moment was calculated by integrating the pressure and viscous forces on the wicket gate surfaces, including the surfaces of the jet channels. Both the beveled trailing edge and blowing from the blunt trailing edge produced approximately a 99% reduction in amplitude of moment fluctuations, which agrees with the findings of Blake et al. [4].
Figure 6. Instantaneous velocity contour plots (in m/s) for the flow in the quasi-2-D periodic distributor vane channel with a blunt trailing edge and a beveled trailing edge.

(a) Blunt trailing edge, no blowing  
(b) Blunt trailing edge, with blowing  
(c) Beveled trailing edge

Figure 7. Unsteady moment on the wicket gate with a blunt trailing edge and a beveled trailing edge. The negative values indicate a closing moment. As the results are from 2-D simulations, they represent a per unit length quantity.

With water injection from the beveled trailing edge a ±5° change in swirl angle was observed at the location of the runner leading edge. This amount of turning was achieved by blowing from each jet independently with a jet speed of 14.9 m/s, or 3% of the inlet flow rate at BEP.

5.2. 3-D Distributor Passage
The distributor passage simulations were used to evaluate the 3-D effects of blowing on the flow distribution entering the runner. Figure 8 shows the change in swirl angle and flow distribution caused by trailing-edge blowing for jet speeds ($V_j$) ranging from 9 to 15 m/s. The inlet volume flow rate to the domain was adjusted by the jet flow rate, in order to achieve a consistent total flow rate through the turbine. The direction of swirl angle change was chosen based on the runner velocity triangles for the corresponding high and low flow/head operating points.

In the low flow case, water was injected from the pressure side jet (Jet 1 in Figure 3). The addition of blowing resulted in an approximately uniform increase in tangential velocity, while the meridional velocity was unaffected, resulting in a uniform increase in swirl angle over the span of the runner inlet. The average swirl angle increased by approximately 2°, 3°, and 4°, for the 9 m/s, 12 m/s, and 15 m/s jet, respectively.
In the high flow case, water was injected from the suction side jet (Jet 2 in Figure 3). The addition of blowing resulted in a uniform decrease in tangential velocity; however, the meridional velocity distribution was altered by the blowing, particularly for the higher jet speeds. Due to the change in meridional velocity, a uniform decrease in swirl angle was not observed, though a reduction in swirl was still achieved for most jet speeds.

5.3. 3-D Runner Blade Passage
The flow through a single blade passage of the runner was simulated at the low flow, BEP, and high flow operating conditions in order to observe the effect of a change in inlet swirl angle on the turbine performance. The computed head and torque were within 4.7%, 1.3%, and 4.3% of the experimental values for the three operating conditions, respectively. Also, the simulations more accurately predicted the turbine performance than Nilsson and Davidson [20].

Due to the segregated nature of these simulations, the change in swirl angle caused by the wicket gate blowing was simulated by modifying the extrapolated experimental inlet velocity profile. The desired change in flow angle was produced by uniformly scaling the inlet tangential velocity profile. For these simulations, the turning angle range was $\pm 6^\circ$.

The resulting runner efficiency for each simulation is shown in Figure 9. By increasing the swirl angle $3^\circ$, the turbine efficiency improved by approximately 4% for the low flow condition. By decreasing the swirl by $3^\circ$, the turbine efficiency improved by approximately 2% for the high flow conditions.

Figure 8. Circumferentially averaged velocity and flow angle profiles, shown at the inlet axis to the runner, for the low flow and high flow operating conditions. Jet 1 is the pressure side jet and Jet 2 is the suction side jet, as shown in Figure 3. All variables are plotted with respect to a normalized curvilinear abscissa $s^*$, where $s^* = 0$ corresponds to the outer surface (band).
flow condition. As expected, any change in swirl angle at the BEP flow rate reduced the turbine efficiency. It should be noted that these efficiencies do not include the losses associated with pumping the necessary amount of fluid to supply the jets. This will be discussed below. As was shown in Figure 8, these values of swirl angle change can be achieved with a jet speed of $V_{j1} = 12 \text{ m/s}$ for the low flow case and $V_{j2} = 15 \text{ m/s}$ for the high flow case.

![Figure 9](image_url)  
**Figure 9.** Change in runner efficiency as a result of modifying the inlet swirl angle of the extrapolated experimental velocity profile. Turbine performance improved significantly by increasing the swirl angle during low-flow operation and by decreasing the swirl angle during high-flow operation.

Though the results of the change in head and torque are not presented here, it can be shown that an increase in swirl angle, at any operating condition, causes an increase in torque according to the Euler Turbomachinery Equation. However, the head increase is more substantial than the torque increase during high flow operation, which results in a decreased efficiency.

### 5.4. Unsteady simulations of the complete turbine

Unsteady simulations of the complete turbine were conducted for BEP, low flow, and low flow with blowing conditions, using a time step corresponding to one degree of the runner rotation per time step. By initializing the flow domain with a coarse grid solution, approximately seven revolutions of the runner were required for the unsteady full-wheel simulation to achieve a time-periodic behavior. For the non-blowing cases, the integrated surface quantities (see Table 1) were then time-averaged over nine complete revolutions of the runner wheel. Only five revolutions were available for the blowing case, as a smaller time step was initially required to maintain a stable solution when the jet was first activated.

Table 1. Comparison of experimental and computed head $H$, torque $T$, and efficiency $\eta$ for the BEP and low flow operating conditions. The results of the steady single runner blade passage simulations are shown for comparison. The percent error from the experimental values are shown in parentheses.

| Data Source         | BEP                  |                      | BEP                  |                      |
|---------------------|----------------------|----------------------|----------------------|----------------------|
|                     | $H$ (m)              | $T$ (Nm)             | $\eta$ (%)           |                      |
| Experiment          | 5.98                 | 388                  | 92.0                 |                      |
| Blade Passage       | 5.97 (-0.2)          | 393 (1.3)            | 94.7 (2.7)           |                      |
| Full-Wheel          | 5.79 (-3.2)          | 378 (-2.6)           | 93.6 (1.6)           |                      |
|                     |                      |                      |                      |                      |
|                     | $H$ (m)              | $T$ (Nm)             | $\eta$ (%)           |                      |
|                     |                      |                      |                      |                      |
|                     | 3.69                 | 170                  | 85.0                 |                      |
|                     | 3.58 (-3.0)          | 178 (4.7)            | 91.5 (6.5)           |                      |
|                     | 3.56 (-3.5)          | 172 (1.2)            | 89.2 (4.2)           |                      |
The results of the full-wheel simulations were within approximately 3% of the measured values for both operating conditions. Compared with the single runner passage simulations, this was an improvement for the low flow condition, but not at BEP. However, the full wheel simulations more accurately modeled the physical operation of the turbine, including the unsteady behavior of the runner rotation, and the full nonaxisymmetric draft tube geometry.

Blowing was introduced at the low flow condition, and a 22% increase in head and a 26% increase in torque were observed. The change in head was evaluated only across the runner wheel, as the addition of mass through the wicket gate jets would skew the results if calculated from the normal measurement locations. This large increase in torque has the potential for increased power production and revenue. As the torque increase was more significant that the head increase, a 2.7% increase in runner efficiency was observed. This is significantly less than predicted by the runner passage simulations, indicating that the downstream presence of the runner strongly affects the overall turning behavior of the jets, and the change in inlet swirl angle was not uniform, as assumed in the runner passage simulations.

Instantaneous surface-streamlines were computed along the runner blade to visualize the secondary flow patterns (see Figure 10). The secondary flows effect the viscous surface forces, and indicate areas of separation or recirculation near the blades. During BEP operation, surface-streamlines initiated on the pressure side of the blade, near the crown, wrapped around to the suction side farther down the span of the blade, showing that a negative relative angle of attack was present at the crown and then transitioned to a slightly positive relative angle of attack when nearing the band.

At low flow, the surface-streamlines near the crown experienced a sharp turn at approximately one-third the chord, which is a clear indication of flow separation in this region. Blowing eliminated the flow separation, and significantly improved the surface flow behavior on the pressure side of the blade. However, the velocity triangles maintained a slightly negative relative angle of attack, and the swirl angle distribution upstream of the runner was still too shallow. Increased amounts of turning will be needed in future work to properly align the velocity triangle with the runner leading edge to produce the maximum improvement in efficiency.

Figure 10. Instantaneous surface-streamlines along the pressure side of the runner blade for the BEP, low flow, and low flow with blowing cases. Surface-streamlines are analogous to oil streaklines. Surface pressure contours are also shown. Images generated in FieldView 13.
5.5. Pumping Requirements to Supply Water to the Jets

An additional pump will be required to supply the jets due to the increased hydrodynamic losses in the jet channels. For low head installations, both the reduced hydrostatic pressure and increased flow rate would further increase the required pumping power. The power necessary to operate the pump must be subtracted from the additional power generated by the turbine.

The actual power required to supply the jet will be dependent on the pump efficiency, losses in the piping network, valves, and filters, as well as losses in the plenum and jet channel design. All these factors must be considered in the final system design. As an initial estimate, the pumping power for the jet system was approximated by the head loss through a representative plenum and jet channel. The plenum was a 7 mm diameter circular cylinder, and the jet channel was a linearly converging duct from 1.0 mm at the plenum to 0.2 mm at the exit. Due to the small channel height, the pressure losses in the channel are likely to dominate all other sources of energy loss, and provide a reasonable pumping power estimate.

The head loss through the plenum and channel \( (h_{\text{loss}}) \) and pumping power \( (\dot{W}_{\text{pump}}) \) are shown in Table 2 for a range of exit jet velocities. The pumping power was approximated as

\[
\dot{W}_{\text{pump}} = N_{\text{wg}} Q_j (h_{\text{loss}} - h_{h_{\text{hs}}}) \rho g
\]

where \( N_{\text{wg}} \) is the number of wicket gates, \( Q_j \) is the flow rate through one jet, and \( h_{h_{\text{hs}}} \) is the additional hydrostatic pressure head at the wicket gate trailing edge.

| \( V_j \) (m/s) | \( Q_j \) (m\(^3\)/s) | \( h_{\text{loss}} \) (m) | \( h_{h_{\text{hs}}} = 0 \) m | \( h_{h_{\text{hs}}} = 3.34 \) m | \( h_{h_{\text{hs}}} = 7.48 \) m |
|----------------|----------------|----------------|-----------------|----------------|----------------|
| 9              | 2.160E-04     | 3.17           | 161             | -8.83          | -219           |
| 11             | 2.640E-04     | 4.14           | 257             | 49.5           | -208           |
| 13             | 3.120E-04     | 5.19           | 381             | 137            | -168           |
| 15             | 3.600E-04     | 6.32           | 536             | 252            | -98.6          |
| 17             | 4.080E-04     | 7.43           | 714             | 393            | -5.15          |

In the full-wheel simulation, a 15 m/s jet increased the runner efficiency by 2.7% at low flow. When including the estimated pumping power, the overall turbine efficiency increased by only 0.9%. Additionally, the calculations show that no additional pumping power is required to supply the jets for the high head operating condition; however, this conclusion may not be accurate. Further analysis is required to fully understand these detailed fluid interactions.

6. Conclusions

To improve off-design hydroturbine performance, the trailing edge of the wicket gate was modified to include a beveled shape with inward inclined jets. A Taguchi method was applied to optimize the position and angle of the jets. The optimal design produced a \( \pm 5^\circ \) change in swirl angle at the inlet to the runner, with a jet speed of 14.9 m/s, or 3% of the inlet flow rate at BEP. 3-D simulations of the distributor passage showed a 4\(^\circ\) maximum increase in swirl angle for the low flow operation. During high flow, additional changes in flow distribution were observed.

The inlet velocity profile for the runner blade passage simulations was modified to achieve a \( \pm 6^\circ \) change in swirl angle. The turbine efficiency was improved by approximately 4% for the low flow condition with a 3\(^\circ\) increase in swirl angle, and by approximately 2% for the high flow condition with a 3\(^\circ\) decrease in swirl angle, neglecting the cost of supplying the jets.
From the results of the full-wheel simulations, the injection of water from the trailing-edge of the wicket gates, using the pressure side jets, successfully increased the swirl angle at the inlet of the runner. For the low flow case, a 15 m/s jet speed, or 2.98% of the inlet volume flow rate, increased the runner efficiency by 2.7%. When subtracting the pumping power necessary to supply the jet, the overall turbine efficiency increased by only 0.9%, which is still a significant improvement in potential revenue for a Francis turbine. The pumping power was estimated by modeling the flow through a representative plenum and jet channel. Improvements to the plenum and jet channel are certainly possible, and should be considered in future work.

Wicket gate trailing-edge water injection significantly improved the pressure and near-wall velocity distribution on the runner blades, and eliminated the major area of flow separation. However, the relative velocity at the runner leading edge was still positioned at a negative angle of attack. Increased amounts of turning should be used in future work to further improve the off-design turbine operation. Additional details for the present work may be found in [21].

Acknowledgments
Funding for the project was provided by the National Defense Science and Engineering Graduate (NDSEG) Fellowship (Award number: FA9550-05-C-0059) and the DOE/PSU Graduate Student Fellowship Program for Hydropower Research (Award Number: DE-EE0002667). Computational time was generously provided by the DOD High Performance Computing and Modernization Program (HPCMP), as part of the NDSEG fellowship.

References
[1] Lewis B J, Cimbala J M and Wouden A M 2012 IOP Conf. Series: Earth and Environmental Science 15 Proc. 26th IAHR Symposium on Hydraulic Machinery and Systems (Beijing, China), IAHRXXVI-121
[2] Aschenbrenner T, Anciger D and Flemming F 2010 Rotor-stator interactions in francis turbines and pump turbines HydroVision International (Charlotte, North Carolina, USA)
[3] Donaldson R M 1956 Journal of Engineering for Power 78 1141–1147
[4] Blake W K 1986 Mechanics of flow induced sound and vibration. (Orlando, Florida, U.S.A.: Academic Press)
[5] Naudascher E 1965 J. Fluid Mech. 22 625–656
[6] Ridjanovic M 1963 Wake with zero change in momentum flux Ph.D. thesis University of Iowa
[7] Wang H 1965 Flow behind a point source of turbulence Ph.D. thesis University of Iowa
[8] Sirviente A I and Patel V C 2000 AIAA Journal 38 613–619
[9] Sirviente A I and Patel V C 2000 AIAA Journal 38 620–627
[10] Brucker K and Sarkar S 2010 J. Fluid Mech. 652 373–404
[11] Chernykh G G, Demenkov A G and Kostomakhac V A 2005 Int. J. Computational Fluid Dynamics 19 399–408
[12] Cimbala J M and Park W J 1990 J. Fluid Mech. 213 479–509
[13] Park W J and Cimbala J M 1991 J. Fluid Mech. 224 29–47
[14] Park W J 1989 An experimental investigation of the turbulent structure in two-dimenstonal momentumless wakes Ph.D. thesis The Pennsylvania State University
[15] Lewis E E 1996 Introduction to Reliability Engineering 2nd ed (Wiley)
[16] Lewis B J, Cimbala J M and Wouden A M 2012 Analysis and optimization of guide vane jets to decrease the unsteady load on mixed flow hydroturbine runner blades Proc. 7th International Conference on Computational Fluid Dynamics (Big Island, Hawaii), ICCFD7-1701
[17] Sottas G and Ryhming I L 1989 3D-Computation of Incompressible Internal Flows. Proceedings of the GAMM Workshop held at EPFL, 13-15 September 1989, Lausanne, Switzerland Notes on Numerical Fluid Mechanics, Volume 39 (Braunschweig/Wiesbaden: Vieweg)
[18] Parkinson E 1995 Turbomachinery Workshop ERCOFTAC II, Test Case 8: Francis turbine
[19] Nilsson H and Davidson L 2003 Int. J. Numer. Meth. Fluids 41
[20] Nilsson H and Davidson L 2002 Validation and investigation of the computed flow in the GAMM Francis Runner and the Hölleforsen Kaplan Runner Proc. 21st IAHR Symposium on Hydraulic Machinery and Systems (Lausanne, Switzerland)
[21] Lewis B 2014 Improving Unsteady Hydroturbine Performance During Off-Design Operation by Injecting Water from the Trailing Edge of the Wicket Gates Ph.D. thesis The Pennsylvania State University Available from https://etda.libraries.psu.edu