Numerical Analysis of the Turbine 99 Draft Tube Flow Field Provoked by Redesigned Inlet Velocity Profiles.

S Galván¹, M Reggio², F Guibault³ and L Castro⁴

¹Mechanical Eng. Dep. Universidad Michoacana de Sán Nicolás de Hidalgo, Apdo. Postal 588 Col Centro, Morelia Mich. Mex., C.P.58001.
²Mechanical Eng. Dep. École Polytechnique de Montréal, P.O. Box 6000, succursale Centre Ville Montréal, QC, Canada, H3C 3A7.
³Computer and Software Eng. Dep. École Polytechnique de Montréal, P.O. Box 6000, succursale Centre Ville Montréal, QC, Canada, H3C 3A7.
⁴Applied Science and Engineering Research Center, Universidad Autónoma del Estado de Morelos. Cuernavaca Morelos, México.

Email: srgalvan@umich.mx, macelo.reggio@polymtl.ca, francois.guibault@polymtl.ca, lauraeg@uaem.mx

Abstract. In recent years, several investigations on hydraulic turbine draft tube performance have shown that the hydrodynamic flow field at the runner outlet determines the diffuser efficiency affecting the overall performance of the turbine. This flow field, for which the principal characteristics are the flow rate and the inlet swirling flow intensity, is mostly developed on turbines designed for low head (high specific velocity) and operated away from their best efficiency point. To identify factors of the flow field responsible for loosing draft-tube efficiency, the correlations between the flow pattern along the diffuser and both swirl intensity and flow rate have been examined. An analytical representation of inlet flow field has been manipulated by a Multi Island Genetic Algorithm through the automatic coupling of multidisciplinary commercial software systems in order to obtain redesigned inlet velocity profiles. This loop allowed determining the profile for which the minimum energy loss factor was reached. With different flow field patterns obtained during the optimization process it was possible to undertake a qualitative and quantitative analysis which has helped to understand how to suppress or at least mitigate undesirable draft tube flow characteristics. The direct correlation between the runner blade design and the kinematics of the swirl at the draft tube inlet should suppose the perfect coupling at the runner-draft tube interface without compromising the overall flow stability of the machine.

1. Introduction
In hydropower plants, the runner which drives the generator, is undoubtedly a key element of the energy conversion process, because the amount of power produced by a turbine is basically equal to the change in angular momentum across this component.

Namely, the power generated by a hydraulic turbine will not only depend on the runner energy conversion, but also on the flow field quality ingested by the draft tube. Theoretically, the flow exiting the runner should have zero swirl [1]; however, this is not achievable in practice and, the power
converted by the runner could be distorted by the flow delivered to the draft tube. In this additional
device, the flow leaving the runner loses its velocity, transforming the excess of kinetic energy into
static pressure. This energy conversion has a significant impact on the overall turbine efficiency and
power, especially for low head (high specific velocity) machines [2] and for machines operating away
from their best efficiency point.

The effects of the inlet swirling flow in a hydraulic turbine draft tube is a very complex
phenomenon, which has been extensively investigated both theoretically and experimentally [3-6].
These investigations have been mainly focused on the effects of the inlet velocity flow on the
inception of draft tube surging phenomenon and on its performance. In more recent years, the
influence of the inlet boundary conditions on the draft tube flow performance has been treated at
length both numerically and experimentally. The principal concern has been the influence of the inlet
swirl flow on the draft tube pressure recovery factor, [7-10].

Among them, [9] presented an important study related to the inlet flow profile optimization. This
researcher observed that to improve the pressure recovery factor required an inlet solid body swirl
with moderate intensity. He tried to find the optimal inlet swirl profile for an existing draft tube
geometry. The parameters to define the solid body swirl profile at the inlet were: the solid body swirl
ratio, the radial component ratio and the axial profile uniformity. It was assumed that the three
components vary linearly in the radial direction. This investigator found the efficiency to be highly
sensitive to all three parameters. This study concluded that the inlet flow profile needed to be
optimized to achieve the best static pressure recovery factor when the existing draft tube cannot be
modified.

Recent investigations [11-14] have revealed the swirling flow structure downstream a runner, by
analyzing experimental data. Axial and tangential velocity profiles downstream a runner have been
matched with analytic profiles given by the same set of equations. More specifically [11] proposed an
outlet runner swirl criteria to avoid an unexpected sudden efficiency drop at certain discharge. This
study included the axial and circumferential velocity components at the runner outlet for 17 operating
points. It was shown that the swirling flow at the runner outlet for a Francis turbine can reasonably be
represented using a superposition of elementary vortices.

Inspired by [9] and [11], it was assumed that an inlet velocity parameterization managed by an
optimization process could build the best field velocity flow for a particular draft tube. This will give
us the opportunity to study different draft-tube flow structures provoked by different inlet velocity
profiles achieving a better understanding of the flow losses.

However, the creation of the optimization process represented a new challenge [15-16], since it was
necessary to reduce the computational time of each simulation, to parameterize the inlet velocity
profiles and to select the objective function to evaluate the draft tube performance.

Then, this work presents as result of the optimization process, a qualitative and quantitative
analysis of the draft tube flow field provoked by redesigned inlet velocity profiles. In order to suppose
the perfect coupling at the runner-draft tube interface without compromising the overall flow stability
of the machine, a direct correlation between the runner blade design and the kinematics of the swirl at
the draft tube inlet has been established. Finally, this analysis has helped to understand how
undesirable draft tube flow characteristics such as secondary flow, irregular evolution, stall and
excessive velocity evolve through this important component of the turbine.

2. Optimization process.
The approach proposed to resolve this numerical optimization can be described in the following steps:
   1. Inlet flow velocity profile parameterization,
   2. Numerical model evaluation,
   3. Optimization algorithm set-up,
   4. Objective function evaluation.
The optimization strategy is built using the iSIGHT software [17], where simulation codes of different disciplines can be coupled. Optimization processes can be configured through a graphical interface with which one can set up, monitor and analyze a design problem. The optimization loop used in this work is shown in figure 1 and the simulations codes are run via shell scripts.

![Flow chart of the draft tube flow optimization process.](image1)

**Figure 1.** Flow chart of the draft tube flow optimization process.

### 2.1. Inlet velocity profiles.

The vortex equations proposed by [11] were established to represent a swirling flow produced by a constant pitch Francis runner. For the current application a Kaplan runner is used instead of a Francis therefore, modifications were applied for a better matching of the velocity equations. Specifically, a near-wall velocity profile and a near-cone hub velocity profile have been added at each curve extremity. To handle the problem of the unknown inlet radial velocity component, a relationship between axial and radial component was used, based on a "geometrical" distribution. In [15-16] the high influence of the radial component on pressure distribution and pressure recovery was demonstrated computationally, in spite of the small magnitude of this velocity component.

Thus, eight parameters were determined by fitting the experimental data and the three velocity components were imposed at section CsIa, as it is shown graphically in figure 2.

![Original inlet velocity profiles at Best Efficient Point.](image2)

(a) Axial (b) Radial (c) Tangential

**Figure 2.** Original inlet velocity profiles at Best Efficient Point.

### 2.2. Draft tube numerical model

The Hölleforsen Kaplan model draft tube, located in Indalsälven Sweden, was used to carry out this study. This geometry, see figure 3, has previously been used in three ERCOFTAC workshops [18-20].

These works present detailed velocity and pressure measurements at a number of measurement sections illustrated in figure 3(a). These measurements are used both to set the correct boundary conditions and to validate the computational results. The model used to obtain reliable numerical data during the optimization process was given by the Navier–Stokes equations. The grid, discretization and turbulence combined choices were discussed through the accuracy of $k–\varepsilon$ turbulence models for the swirling flow in the Turbine 99 draft tube [21]. Discussion was based on graphical results and by comparing numerical simulations and experiments in the operational mode T (best efficiency point).

A perspective view of the draft tube is shown in figure 3(b) which presents a vertical symmetrical plane and six cross section planes where the results obtained in this work will be presented.
2.3. Optimization algorithm

In the present work, the optimization approach is based on an efficient Genetic Algorithm (GA) technique called Distributed GA (DGA). This method has been chosen because it is effective to seek an optimum solution in a wide design space. However, optimizations based on GA need a large number of evaluations making them better suited for a parallel computing system. This exploratory technique is established within the iSIGHT software as the Multi-Island Genetic Algorithm (MIGA). The major feature distinguishing a MIGA approach from a traditional GA is the fact that each population of individuals is divided into sub-populations called islands. The usual genetic operations (selection, reproduction and mutation) are performed separately on each island. A further operation called migration is used to transfer some individuals from an island to another. The migration process is controlled by two major parameters which are the rate of migration and the interval of migration. A parametric study was made by [22] to start an optimization loop with a high chance of success avoiding extensive preliminary sensitivity analysis proving also to be well suited for solving highly nonlinear problems, like the present one.

2.4. Objective function

Global performance quantities including the loss coefficient factor $\zeta$ presented in equation (1), wall pressure recovery factor $C_{pw}$ in equation (2) and the mean pressure recovery factor $C_{pm}$ in equation (3) were tested to select the most appropriate objective function [22]. This study revealed that $\zeta$ is highly sensitive to the changes of each inlet velocity profile parameters of each equation given by [11].

$$\zeta = \frac{1}{A_{in}} \int_{in} P dA - \frac{1}{A_{out}} \int_{out} P dA$$

$$\frac{1}{2} \rho \left( \frac{Q}{A_{in}} \right)^2$$

Where $P_f = P + 0.5 \rho (u^2 + v^2 + w^2)$. The total loss in the flux, normalized with an energy flux estimator given by the denominator, is independent of the swirl component at the inlet.

The wall pressure recovery coefficient $C_{pw}$ given by equation (2), is based on wall pressure considered at different points on the wall where $P_{out:wall}$ is the averaged static wall pressure across the outlet section and $P_{in:wall}$ is the averaged static wall pressure across the inlet section.

$$C_{pw} = \frac{P_{out:wall} - P_{in:wall}}{\frac{1}{2} \rho \left( \frac{Q}{A_{in}} \right)^2}$$

In equation (3), the mean pressure recovery $C_{pm}$, is based on the mean values of the static pressure over the inlet and outlet areas.

$$C_{pm} = \frac{1}{A_{out}} \int_{out} P dA - \frac{1}{A_{in}} \int_{in} P dA$$

$$\frac{1}{2} \rho \left( \frac{Q}{A_{in}} \right)^2$$
The loss coefficient factor $\zeta$ has also the benefit of having no restriction on the inlet flow. Thus the optimization algorithm should find the best results in a wide range of normalized inlet flows, which are defined by means of equation (4).

$$Q_{\text{nor}} = \frac{Q}{Q_{\text{ref}}}$$  \hspace{1cm} (4)

And the reference quantity for monitoring the numerical solution is given by the difference of mass, equation (5).

$$\Delta m = m_{\text{inlet}} - m_{\text{outlet}}$$  \hspace{1cm} (5)

$S$, equation (6), corresponds to the swirl number defined as the axial flux of swirl momentum divided by the axial flux of axial momentum.

$$S = \frac{\int_0^R (\rho V_r^2)(rV_r)dr}{R \int_0^R (\rho V_r^2)(rV_r)dr}$$  \hspace{1cm} (6)

Other engineering quantities of interest are the kinetic energy correction factors, one related to the axial velocity $\alpha_{ax}$, in equation (7).

$$\alpha_{ax} = \frac{1}{A_o u^t} \int_A u^3 dA$$  \hspace{1cm} (7)

And the other to the swirl velocity $\alpha_{sg}$, in equation (8).

$$\alpha_{sg} = \frac{1}{A_o u^t} \int_A w^3 udA$$  \hspace{1cm} (8)

Physically $\alpha$ represents the ratio of the actual kinetic-energy flux, at a given cross section of an internal flow stream, to the minimum kinetic-energy flux which could exist at the particular flow rate.

The momentum correction factor, equation (9), also called momentum coefficient or Boussinesq coefficient is the momentum of water passing through the diffuser. It is defined as the ratio of momentum of the flow per second based on actual velocity to the momentum of the flow per second based on average velocity across the section. It is denoted by:

$$\beta = \frac{1}{A_o u^t} \int_A u^3 dA$$  \hspace{1cm} (9)

3. Optimization process results.

The objective function behavior with respect to each evaluation, the swirl intensity $S$ and the flow rate $Q_{\text{nor}}$ are presented in figure 4. The objective function is plotted for the best individual of each generation versus the index of iteration, figure 4(a). This graph indicates that the convergence has been reached with $\zeta = 0.29\%$ after $10 \times 3 \times 100 = 3000$ runs. Figure 4(b) shows the variation of the objective function with respect to the inlet swirling flow intensity along the optimization process. The swirl intensity was reduced 96% of its original value when the objective function was minimized (see figure 4(b). Figure 4(c) presents the variation of the mass flow rate when the objective function is minimized. Since there is no restriction to inlet mass flow, it is observed that the optimization algorithm executed the entire global search at overflow. On this account, the mass flow achieves the value of $Q_{\text{nor}} = 1.86$ which means that the optimized flow rate point should be out of the Best Efficiency Point of the turbine (BEP). Since there is only a single swirl number for which the objective function is minimized, it appears that the ratio of axial to tangential component of velocity is what mostly affects the energy loss factor, which is only very slightly affected by the flow rate. This allows scaling the mass flow without impacting the draft-tube performance (see Table 1).
4. Draft Tube Flow Analysis.

Figure 5 presents three different inlet velocity profiles obtained from the runs 256, 643 and 2,966 of the optimization process which were selected as boundary conditions to develop the quantitative and qualitative flow analysis. The velocity profiles selected provoke an important step or gradient decrease of the objective function \( \zeta \), one order of magnitude, as is shown in Table 1. The principal characteristic of these profiles with respect to the original is the near wall peak of the axial and radial components and the change of direction in the radial distribution of the tangential velocity.

All these profiles were obtained at different mass flow rates as is shown in Table 1. In order to match the mass flow rate at the BEP, the inlet velocity profiles were scaled, maintaining the same swirl number. The \( \zeta \) value for both flow rates is very similar and the flow mass imbalance of the scaled profiles indicates a good solution of the CFD simulation. Figure 5 also presents these new profiles which maintain the same shape.

Table 1. Engineering quantities reached by the selected inlet velocity profiles.

| Run     | \( Q_{nor} \) | \( \zeta \) | \( S \) | imb  |
|---------|---------------|-------------|-------|------|
| original| 1.0           | 0.1755      | 0.2600| 2.58\times10^{-6} |
| 256     | 1.8031        | 0.1291      | 0.0694| 1.45\times10^{-6} |
|         | 1.0000        | 0.1217      | 0.0694| 3.22\times10^{-6} |
| 643     | 1.9738        | 0.0663      | 0.0759| 4.50\times10^{-6} |
|         | 1.0000        | 0.0674      | 0.0759| 9.00\times10^{-6} |
| 2966    | 1.8613        | 0.0038      | 0.0129| 3.00\times10^{-6} |
|         | 1.0000        | 0.0042      | 0.0129| 2.01\times10^{-6} |

4.1. Quantitative analysis

Figure 6 presents the comparisons between the pressure recovery and energy loss along the draft tube sections provoked by the inlet velocity profiles selected After this zone, the inlet velocity profile has the capacity to increase the pressure, maintaining a low energy loss factor along the draft tube
reducing the hydraulic energy loss by a factor of 78% compared with the original flux. The mean pressure recovery factor, figure 6(b), and the wall pressure recovery factor, figure 6(c), do not present the same performance order generated by the optimized flow, because the optimization was based on the energy loss factor, but even with the $C_{p_m}$, which has shown a poor sensitivity, the efficiency increment is significant reaching 5.6%.

Figure 7 presents the draft tube performance evaluated along the draft tube by considering equations 6-9. Figure 7(a) shows the effect of the non-uniformity in the velocity profile by the axial kinetic energy factor $\alpha_{ax}$. Since diffusion requires a reduction in kinetic energy flux, any increase in the non-uniformity represents an increase of $\alpha_{ax}$. The optimized profiles reduce this factor at the downstream section $Cs4a$. The amount of kinetic energy of the tangential velocity component is quantified in figure 7(b) and represents the augmentation of actual kinetic energy due to swirl. The optimized profiles achieve a reduced number along all the draft tube length due to sufficient diffusion provoked by a lower swirl number. Figure 7(c) presents the momentum correction factor which is the momentum of water passing through the diffuser and it is different from the unity when the velocity across a flow area is not uniform. The optimized flow shows a better approximation to this value at the end of the draft tube. Figure 7(d) presents the swirl intensity along the diffuser. For the optimized flow, this quantity is maintained almost constant throughout the draft tube except at the end, where it undergoes a slight increase. On the contrary, the other flows suffer a drop of the swirl intensity at the end that provokes a wall flow detachment.

4.2. Qualitative analysis

Figure 8 presents the total pressure contours provoked by the original flow and the optimized ones, at the draft tube symmetry plane. Stronger decrease of the total pressure zone is obtained by the optimized profiles at the $Cs1b$ section, figure 8(d). The pressure reduction below the runner hub zone indicates back-flow mean velocity. The radial and axial velocity gradients near the wall raise the pressure on the wall, avoiding flow separation and consequently low pressure regions beneath the runner hub, which decreases considerably in the optimized flow. This allows a homogeneous pressure
distribution downstream, because there is no adverse pressure gradient in the elbow zone. As such, the flow generated by the optimized profiles affect the draft tube function through a blockage effect.

![Image](image1)

**Figure 8.** Total pressure contours at the draft tube symmetry.

The streamlines and vectors at several cross sections are shown in figure 9 for the original and optimized inlet flows. The picture reveals asymmetric structures in the secondary flow, due to the change of rotational direction at the inlet. While the tangential velocity component exists in all the original sections, in the optimized sections this component is reduced up to practically disappearing except for a strong radial component towards the cone wall. The streamlines in the sections \(CS2\) and \(CS3\) indicate two counter-rotating vortices on each side of the centerline. The left vortex seems to be stronger than that of the right side. For the optimized flow, the counter-rotating vortices have a lower intensity and they are finally displaced to the lower wall. Almost the same flow pattern continues downstream along the next sections \(CS4a\), \(CS4b\) and the outlet. At the outlet section, both the original and optimized flow maintains the secondary flow with a high intensity vortex zone in the upper right corner.

![Image](image2)

**Figure 9.** Streamlines and vectors at draft tube sections provoked by the inlet velocity profiles.

Figure 10 presents the axial velocity contours at different draft tube cross sections. At the inlet section \(CS1a\), the principal difference among axial velocity profiles lies in the radial distribution of its magnitude. At section \(CS1b\), the axial velocity magnitude has been reduced, however, in the original profile, there is a large area of back flow downstream of the rotating hub. Inversely, for the optimized profiles this area is progressively reduced. At the downstream sections, the picture reveals a reduction of the main flow towards the left side, in the original flow, with back-flow at the lower wall. In the optimized flow, figure 10(d), the zone with no main flow agrees with the vortex zone presented in
figure 9(d), whereby it is argued that back flow is not present at the outlet section of the optimized flow.

![Figure 10](image)

Figure 10. Axial velocity contours at the draft tube sections provoked by the inlet velocity profiles.

Figure 11 shows the streamlines and vectors at the symmetry mid-plane from draft tube inlet to outlet. Due to the changes of rotation direction along the inlet radius and the near-wall axial velocity increase for the optimized profile, as is presented in figure 5, the main flow moves to the lower wall and consequent bend blockage and back-flow beneath the hub are suppressed. In these figures it can be appreciated that the main velocity uniformity is increased by the optimized flows, and the back-flow at the outlet section has been suppressed.

![Figure 11](image)

Figure 11. Streamlines and vectors at the draft tube symmetry provoked by inlet velocity profiles.

4.3. Flow angle

The direct correlation between the runner blade design and the kinematics of the swirl on the draft tube inlet is established by the relative flow angle, [16]. An important property of the swirling flow downstream a constant pitch hydraulic turbine runner is that the relative flow angle depends only on the blade exit angle provided that the flow remains attached. The significant changes in the circumferential and axial velocity profiles in the survey section can be associated to the upstream variation of the blade trailing edge as it is shown in figure 12. The flow angle seems to fit the
experimental data and the blade trailing edge will not suffer an important modification with regard to the original blade angle. This parameter could be used as a quantitative correlation between the swirling flow sought at the draft tube inlet and blade shapes that can be easily produced.

5. Conclusions

In order to improve the runner-draft tube coupling, an optimization process of the velocity profiles at the draft tube inlet was developed using numerical tools. This process allowed determining the profile for which the minimum energy loss factor was reached. Three theoretical inlet velocity profiles generated by a Kaplan runner were selected to study the qualitative and quantitative flow structure they provoke. The results obtained clearly illustrate the importance of the flow uniformity at the end of the conduit. If the momentum parameter can be reduced in the last draft tube sections, the draft-tube performance is increased considerably. Another important aspect is the reduced blockage area produced by the optimized inlet velocity profile. This blockage promotes an adverse pressure gradient in the cone, deteriorating the pressure recovery effect. Thus, the drop in energy loss is explained by reduced global machine stability. Also, the flow angle could be used as a quantitative correlation between the swirling flow sought at the draft tube inlet and blade shapes. This could filter non realistic runner blades during a distinct optimization process. Finally, it can be stated that the results of this optimization methodology helped us understand how the inlet flow characteristics can be changed in order to suppress undesirable effects such as secondary and back flow, irregular evolution and excessive swirling intensity along the draft tube.

References

[1] Falvey H 1993A primer on draft tube surging Hydro Review (February) 77–86.
[2] Tridon S, Barre S, Ciocan G-D and Tomas L 2010 Experimental analysis of the swirling flow in a Francis turbine draft tube: Focus on radial velocity component determination European journal of Mechanics B/Fluids 29 321–335.
[3] Falvey H and Cassidy J 1970 Frequency and amplitude of pressure surges generated by swirling flows InProceedings of the International Association for Hydraulic Research. Symposium on Hydraulic Machinery and Cavitation Stockholm, Sweden, No. E1 1–12.
[4] Padle U 1974 Model and prototype turbine draft tube surge analysis by the swirl momentum method InProceedings of the International Association for Hydraulic Research. Symposium on Hydraulic Machinery and Cavitation. Stockholm Sweden No. III 1–13.
[5] Monacelli G and Cooper P 1987 Performance of draft tubes for small-hydropower turbine InWater Power’87 1200-1209 ASCE.
[6] Kubota T, Han F and Avellan F 1996 Performance analysis of draft tube for GAMM Francis turbine InProceedings of the XVIII IAHR Symposium, Hydraulic Machinery and Cavitation.Dordrecht NL No. 1 130–139.
[7] Shyy W and Braaten M 1986 Three-dimensional analysis of the flow in a curved hydraulic turbine draft tube International Journal of Numerical Methods in Fluids 6 861–882.
Vu T and Shyy W 1990 Navier-Stokes flow analysis for hydraulic turbine draft tubes. Journal of Fluids Engineering 112:199–204.

Vu T 1989A design parameter study of turbine draft tube by viscous flow analysis. WATERPOWER 89:557–566.

Mauri S, Kueny J, and Avellan F 2002 Werlé-Legendre separation in a hydraulic machine draft tube. In ASME 2002 Fluids Engineering Division Summer Meeting, Montréal, Québec Canada.

Susan-Resiga R, Ciocan G, Anton I and Avellan F 2006 Analysis of the swirling flow downstream a Francis turbine runner. Journal of Fluids Engineering 128:177–189.

Gagnon J, Iliescu M, Ciocan G, and Deschênes C 2008 Experimental investigation of runner outlet flow in axial turbine with LDV and stereoscopic PIV. In 24th IAHR Symposium on Hydraulic Machinery and Systems, Foz do Iguassu, Brazil, October 27–31.

Muntean S, Ciocan T, Susan-Resiga R, Cervantes M and Nilsson H 2012 Mathematical, numerical and experimental analysis of the swirling flow at a Kaplan runner outlet. In IOP Conference Series: Earth and Environmental Science Vol 15 IOP Publishing p. 03 2001.

Susan-Resiga R, Muntean S, Avellan F and Anton I 2011 Mathematical modelling of swirling flow in hydraulic turbines for the full operating range. Applied Mathematical Modelling 35(10):4759–4773.

Galván S, Reggio M and Guibault F 2012 Optimization of the inlet velocity profile in a conical diffuser. In Proceedings of the ASME 2012 Fluids Engineering Summer Meeting Volume 1 Symposia Parts A and B, Rio Grande, Puerto Rico, USA July 8–12 Paper No. FEDSM2012-72103:125-134.

Galván S, Reggio M and Guibault F 2013 Inlet velocity profile optimization of the turbine 99 draft tube. In Proceedings of the ASME 2013 Fluids Engineering Summer Meeting Volume 1A Incline Village, Nevada, USA July 7-11 Paper No. FEDSM2013-16473 V01AT02A007-FEDSM2013-16473.

isight 1994 iSIGHT 9.0.1 Engineous Software Inc.

Gebart B R, Gustavsson L H and Karlsson R I (Editors) 2000 Proc. of Turbine-99 – Workshop on draft tube flow (Porjus, Sweden, 20-23 June 1999) Luleå University of Technology, Technical Report 2000:11.

Engström T F, Gustavsson L H and Karlsson R I (Editors) 2001 Proc. of Turbine-99 – Workshop 2 (Ålvkarleby, Sweden, 18-20 June 2001) Luleå University of Technology.

Cervantes M J, Engström T F, and Gustavsson L H (Editors) 2005 Proc. of Turbine-99 III. (Porjus, Sweden 7-9 December 2005) Luleå University of Technology, Technical Report 2005:20.

Galván S, Reggio M and Guibault F 2011 Assessment study of k–ε turbulence models and near-wall modeling for steady state swirling flow analysis in draft tube using fluent. Eng. Appl. Comput. Fluid Mech 5(4):459–478.

Galván S, Rubio C, Pacheco J, Mendoza C, and Toledo M 2013 Optimization Methodology Assessment for the Inlet Velocity Profile of a Hydraulic Turbine Draft Tube. part I: Computer Optimization Techniques. Journal of Global Optimization 55(1):53–72.