Design of a Kaplan turbine for a wide range of operating head
-Curved draft tube design and model test verification-

Pohan KO\(^1\), Kiyoshi MATSUMOTO\(^2\), Norio OHTAKE\(^3\), Hua DING\(^4\)

\(^{1,2,3}\) Toshiba Corporation
20-1 Kansei-cho Tsurumi-ku Yokohama, 230-0034, Japan
\(^4\) Toshiba Hydro Power (Hangzhou) Co., Ltd
1 Gongren Rd., Fuchunjiang Tonglu, Zhejiang. China

E-mail: pohan.ko@toshiba.co.jp, kiyoshi1.matsumoto@toshiba.co.jp,
norio.ohtake@toshiba.co.jp, ding.hua@toshiba-thpc.com

Abstract. As for turbomachine off-design performance improvement is challenging but critical for maximising the performing area. In this paper, a curved draft tube for a medium head Kaplan type hydro turbine is introduced and discussed for its significant effect on expanding operating head range. Without adding any extra structure and working fluid for swirl destruction and damping, a carefully designed outline shape of draft tube with the selected placement of center-piers successfully suppresses the growth of turbulence eddy and the transport of the swirl to the outlet. Also, more kinetic energy is recovered and the head lost is improved. Finally, the model test results are also presented. The obvious performance improvement was found in the lower net head area, where the maximum efficiency improvement was measured up to 20\% without compromising the best efficiency point. Additionally, this design results in a new draft tube more compact in size and so leads to better construction and manufacturing cost performance for prototype.

The draft tube geometry parameter designing process was concerning the best efficiency point together with the off-design points covering various water net heads and discharges. The hydraulic performance and flow behavior was numerically previewed and visualized by solving Reynolds-Averaged Navier-Stokes equations with Shear Stress Transport turbulence model. The simulation was under the assumption of steady-state incompressible turbulence flow inside the flow passage, and the inlet boundary condition was the carefully simulated flow pattern from the runner outlet. For confirmation, the corresponding turbine efficiency performance of the entire operating area was verified by model test.

1. Introduction
As for balancing the power consumption demands, hydro power functions as a regulator reacting on the load fluctuation throughout day and night. As the power grid load fluctuates smoothly and gently over a long period of time, conventional hydro turbines are sufficiently doing great job over decades. But as for modern day, the power usage becomes much more complex and unpredictable over the booming of individual energy storage devices, transportations, as well as the commencement of renewable energy of other kinds for reducing carbon footprint. As a result, the grid load fluctuates drastically in a relatively short period of time. Thousands of megawatt change may occur simply in hours. Furthermore, the climate change upsets the global water circulation system and the swing of
the river water level becomes unpredictable. For working as a secure renewable energy, the
development of the new hydro power involves seeking the way to become more flexible in a wider
operating power and head range. For top high head conditions, Pelton type hydro turbine gets the
spotlight for its full range operability, but when it comes to the lower water net head, the limit of the
geographical environment kicks in, so Francis and the axial type turbines are still highly in demand.
Therefore the corresponding researches and developments for expanding operating range are eager to
be done.

Aside from hydro turbine, wide operating range design for other turbomachine is also often a
challenging research topic as the off-design points involve unclearly understood and complex fluid
behaviours. The physics phenomena as well as the prediction and measurement methods have been
widely studied and discussed. As turbomachines are operating in off-design conditions, the streams of
the working fluid misalign with turbo blade’s designed angle, camberline, as well as the shape of other
parts of the flow passage. For this reason, flow separation, vortices, as well as other kinds of
secondary flow take place. These flow behaviours are complex and inducing noise and pressure
pulsation, and then consequently deteriorate machine’s performances and even structure. Furthermore,
these flow behaviours and energy may flow down to the blade wake. As for turbomachines, if the
instability of the wake shedded from the rotating blade gets transported, the hydraulic problems are
induced to confine the operating range. As for common compressors and pumped machines, the wake
flow induced secondary flow often occurs among blades, and even between the rotating and stationary
blades to compromise the turbine performance in off-design operations [1, 2]. As for hydro turbine,
the secondary flow involved blade wake often sheds and curls up to the downstream to induce
turbulence eddy and swirl to the draft tube. This deteriorates the hydraulic performance in the off-
design conditions at the full load [3, 4, 5, 6] as well as the part load [7, 8].

As for eliminating swirl in the draft tube for enhancing off-design performances to achieve a wide
operating range, splitter runner with outlet optimization [9] was one of the approaches for Francis type
turbine of higher head machine. For those turbines fitted to lower head, the introduction of foreign
structures [8, 10] and water jet [11] was part of the successful solutions. In this paper, medium head
Kaplan type turbine is focused. For the reasons of 1) Kaplan type turbine has rotatable blades and no
structural possibility for applying splitter runner application, 2) as for low head machine, the draft tube
is often the key portion affecting the entire turbine performance over its effect recovering the runner
outlet kinetic energy back to static pressure, and 3) the draft tube loss itself is potentially up to 50% as
flow rate is high [12]. Therefore, the design and study in this paper are narrowed down to the draft
tube. The draft tube is designed and analysed by numerical simulation and then verified by model test.
The new design is found successfully functioning on significantly expanding operating head range
without best efficiency drop. Furthermore, no additional help of foreign structures, water jet, and
aeration is applied.

2. Prediction method of model turbine performance

2.1. Fundamental equations and Solver
A commercial CFD software “ANSYS CFX” is used. Reynolds averaged Navier-stokes (RANS)
equations shown as below are the governing equation solved for steady state incompressible
turbulence flow behavior prediction.

\[
\frac{\partial \overline{\mu}}{\partial x_i} = 0 \tag{1}
\]

\[
\rho \overline{u_j} \frac{\partial \overline{\mu}}{\partial x_j} = \frac{\partial}{\partial x_j} \left( -\overline{p \delta_{ij} + \mu \overline{S_{ij}} - \rho \overline{u_i u_j}} \right) \tag{2}
\]

\( \overline{u_i} \) designates velocity components, \( p \) designates pressure, \( x_i \) designates Cartesian coordinate
components whereas the over bar designates the time-averaged value. \( \rho \) designates flow density and
\( \mu \) designates the laminar dynamic viscosity of the flow. \( \delta_{ij} \) is the Kronecher delta, \( S_{ij} \) is the strain rate tensor. \(-\rho u_i u_j\) is defined as Reynolds stress term which summarizes the time-fluctuating turbulence flow behavior. As for predicting the turbulence eddy viscosity for mathematically modeling \(-\rho u_i u_j\), Boussinesq assumption, equation (3), as well as standard k-\( \varepsilon \) and original k-\( \omega \) turbulence two-equation models, equation (4), are applied, and then the Reynolds stress term is expressed as follow.

\[
-\rho u_i u_j = 2\mu_T S_{ij} - \frac{2}{3}\rho k \delta_{ij}, \quad \text{also} \quad -\rho u_i u_j = \mu_T \Omega_{ij}
\]

(3)

\[
\mu_T = \rho C_\mu \frac{k^2}{\varepsilon} \quad \text{and} \quad \mu_T = \rho k/\omega
\]

(4)

\( \Omega_{ij} \) is the shear velocity gradient. The turbulence kinetic energy \( k \), dissipation rate \( \varepsilon \), and dissipation specific rate \( \omega \) are obtained by solving Menter’s SST k-\( \omega \) model [13], which applies 1) a zonal weighting blending function \( F_1 \) to describe k-\( \varepsilon \) in k-\( \omega \) formation for achieving k-\( \omega \) for inside and k-\( \varepsilon \) model for outside of boundary layer treatments together, 2) an additional limiter, equation (5), of another blending function \( F_2 \) for counting in shear stress transport effect to avoid \( \mu_T \) from being over-predicted as adverse pressure gradient flow occurs inside boundary layer. This helps accurately capturing the flow separation phenomena [14], which is also often seen in the draft tube.

\[
\mu_T = \rho \frac{a_i k}{\max(a_i \omega, \Omega_{ij} F_2)}
\]

(5)

where \( F_2 = 0 \) for free shear zone and \( F_2 = 1 \) for boundary layer zone. \( a_i \) is the model constant. This limiter keeps the original \( \mu_T = \rho k/\omega \) for the free shear flow whereas applies Bradshaw’s assumption by using \( \mu_T = \rho a_i k/\Omega_{ij} \) to achieve \(-\rho u_i u_j \propto k \) for flow the inside the boundary layer. This gains high accurate results for engineering applications [15].

2.2. Computational grid and boundary condition
A medium head Kaplan type model hydro turbine of ns450 is studied. Although draft tube is focused in this study, prior to which, the entire flow passage of the targeted Kaplan type turbine are numerically simulated in various operating conditions for extracting the flow patterns at the runner outlet. These are then applied as the inlet boundary conditions for draft tube design and analysis.

Configuration and summery of this turbine are shown in Figure 1. The picked-up calculation conditions for draft tube design are summarised in Table 1. As for designing a turbine system of wider operating range, beside the best efficiency point (BEP) the calculation conditions involve other three off-design points of different flow discharges and water net heads.

![Figure 1. Configuration of ns 450 Kaplan model turbine](image)

| Specific speed (ns) | 450 (min⁻¹,m-kW) |
|---------------------|------------------|
| Runner outlet diameter (De) | 350 mm |
| Stay vane number | 28 |
| Guide vane number | 28 |
| Runner blade number | 5 |
Table 1. Calculation conditions

| Operating point number | Runner blade angle [°] | Guide vane opening [mm] | Rotation speed [rev/min] | Flow rate [lit/s] |
|------------------------|------------------------|-------------------------|--------------------------|------------------|
| OP1                    | 16                     | 23.1                    | 1455                     | 357              |
| OP-BEP                 | 22                     | 25.7                    | 1255                     | 472              |
| OP2                    | 32                     | 28.3                    | 1255                     | 664              |
| OP3                    | 30.9                   | 30.9                    | 1455                     | 698              |

P.S. Runner rotates in clockwise direction from the top view.

The computational model is shown in Figure 2. The number of grid points is about 30 million for the entire flow passage and about 0.3 million for draft tube. In the numerical simulation, it adopts the high resolution numerical scheme for the advection term of the transport equations. Therefore high quality grid distribution is prepared for maintaining the algorithm stability. As for accurate inlet boundary condition for draft tube simulation, an automatic multi-block grid generation method minimizing the skewness of the block and cell is applied to the turbine runner. [16] Opening pressure and direction with 0 Pascal of relative pressure is applied as the outlet boundary condition. The non-slip boundary condition is prescribed to the flow passage walls.

Figure 2. Overview of computational model (entire passage & draft tube)

3. Model Test

The hydraulic performance of the draft tube is measured and verified on a test rig at the Hydraulic research laboratory in TOSHIBA HYDRO POWER (HANGZHOU) CO., LTD in China. The model test stand in China shares the same test rig structure in TOSHIBA Japan as shown in Figure 3. The model test follows IEC standard. The hydraulic performance data were measured with existing base model and the inaccuracy of measurement has been estimated less than ±0.25%.

(a) model test stand in TOSHIBA China  (b) model test rig structure in TOSHIBA Japan

Figure 3. Model test stand for model test
4. Design results and discussion

4.1. Design Concept

As for recovering the runner outlet kinetic energy back to static pressure, straight diffusors were first introduced in 19th century. As the inquired machine capacity and size grew with the market demand, the shape of the draft tube was altered and unavoidably compromised. The diffusor was then curved for fitting in shallower excavation for achieving more competitive construction and manufacturing cost performance. This additional curved part, also known as the elbow, has drawback bringing down the hydraulic performance especially in off-design conditions to confine the operating range. Therefore, the major shape parameters for the design in this study are chosen to be placed at the elbow part of the draft tube. The parameters include elbow curvature, height, width, lateral tilting angle. As for the diffuser part, the shape is correspondingly and linearly adjusted to ensure a smooth outline and area change with elbow. There are 8 planar sections for the elbow. They forms circle in shape from the draft tube inlet, to radius rectangular in shape connecting the diffusor which involves another 4 radius rectangular planar sections. Also, as for having a substantial design limiting the cost of excavation and concreting for prototype, the width and the depth are consciously constrained. Finally, carefully designed number and position of center-piers are decided for hydraulic performance and structure strength enhancements.

Because the starting point of the design in this study bases on an existing draft tube, the typical Swedish parameterization step is not necessary.

4.2. Numerically Simulated performance

As for concerning that the swirl transportation inside the draft tube works as one of the main factors affecting the turbine performance, the contour of the absolute velocity and swirl strength $m$ defined as equation (6) are numerically predicted. The results are illustrated in Figure 4 and Figure 5, respectively. They are also referred for estimating the draft tube in the designing process.

$$m = \frac{\int r V_\theta V_z \, dA}{R \int V_z^2 \, dA}$$

where $V_\theta$ and $V_z$ respectively designate axial and tangential velocity whereas $r$ designates the radial distance with respect to the direction normal to the corresponding planar section. $dA$ is the infinitesimal area of the planar section. $R$ designates the characteristic radius of the planar section.

Given that the draft tube is introduced for recovering the runner-induced kinetic energy to static pressure energy, the pressure recovery rate $C_p$, which is defined as equation (7), along the draft tube is calculated as results in Figure 6. It is also key for hydraulic performance estimation.

$$C_p = \frac{\bar{P}_{\text{outlet}} - \bar{P}_{\text{inlet}}}{\rho Q (A/A_{\text{inlet}})^2}$$

where $Q$ designates flow discharge, $\bar{P}$ designates mass flow rate averaged static pressure, $A$ designates the surface area while the subscript $\text{inlet}$ designates the location of the draft tube inlet.

Finally, the hydraulic loss distribution along the draft tube is predicted from the numerical results and is illustrated in Figure 7. The curves plot the piled-up hydraulic loss throughout each planar section from the inlet to the outlet. The sectional loss is predicted by the flow rate averaged total pressure difference between two planar sections.

Seen from Figure 4, it first illustrates that the new draft tube features with 1) narrower width and lower height of planar section connecting elbow and diffusor, 2) shorter but one additional center-pier for maintaining the structure strength. Aside from the hydraulic performances, these guarantee a better construction cost performance. By seeing the velocity contours in overall operating conditions, the
new draft works better on balancing out the lateral velocity distribution at the elbow. Also, the new draft induces slower flow at the outlet and so is proven eliminating more kinetic energy. As for the old draft operating in the three off-design conditions, clear swirl-like structure is observed at the left side of the diffusor whereas seen much weaker in the new draft. In Figure 5, swirl strength jump occurs at the elbow of new draft, but it is immediately suppressed below the old draft at the diffusor part in all operating conditions.

![Figure 4. Contour of velocity](image)

![Figure 5. Comparison of normalized swirl strength](image)
In Figure 6 and Figure 7, the pressure recovery rate and hydraulic loss comparisons are illustrated. Aside from the operating condition of OP-BEP, the new draft has better pressure recovery rate at the elbow in off-design operating conditions although an obvious drop appears before flow enters the center-pier sections. This higher pressure recovery region also has the hydraulic loss suppressed at the elbow. As a result, even though the pressure recovery rate in OP1 condition is worse at the diffusor part, the loss increment does not override that of the existing old draft tube. As for OP3 of lower net head with bigger discharge, both pressure recovery and loss performance are better. As for OP-BEP of designed condition, no positive effect is observed possibly due to the trade-off often seen in turbine design. Despite of this, for overall, the new draft tube is numerically estimated to have the potential for widening the operating range.

Figure 6. Comparison of normalized pressure recovery rate

Figure 7. Comparison of normalized hydraulic loss
4.3. Model tested performance
As for the final stage, the hydraulic performance of the new draft tube has been verified by model test. The model test has been carried out for entire operating conditions to ensure the operating range is successfully expanded after applying the new draft tube. The turbine efficiency curves are shown in Figure 8. Three runner blade openings of 15°, 25°, and 32° representing the part load, designed point, and full load, respectively, have been chosen and tested with five guide vane openings for each. The runner rotating speed is swung to ensure the water net head covering \( n_{ED} \) from 0.5 to 1.2. The resulted performance curves are plotted with the dashed curves of existing old draft tube for clear comparison. From these curves, they clearly show that although slightly efficiency drop is measured at \( n_{ED}=0.5 \), a drastic efficiency improvement is measured at \( n_{ED}=1.2 \) whereas the best efficiency is kept. The same tendency is measured and observed in part load, design point, and full load operating conditions. As a result, this proves a successful design of a draft tube for widening operating head range.

![Figure 8. Comparison of the model turbine efficiency curves](image)
5. Conclusions
A draft tube has been numerically designed and experimentally proved functional on expanding the operating head range without the need of extra structure or working fluid no matter the change of discharge condition. Although the shape parameter for draft tube outline design is focused on the elbow, an obvious hydraulic performance improvement is obtainable. The new draft tube is summarized as follows:

(1) As for minimizing the number of shape design parameter as well as for a substantial design concept study for prototype, elbow part is focused due to its drawback on hydraulic performance, as well as its direct impact on construction and excavation-related cost performance. As a good result, the new draft tube has a narrower planar section connecting the elbow and diffusor parts, and has the elbow of smaller curvature and a diffusor of bigger expansion rate.

(2) As for the numerical results, the new draft tube works successfully, especially in off-design conditions, on suppressing the hydraulic loss growth, as well as on giving quick rise to pressure recovery at the elbow. Also, it works successfully on suppressing the kinetic energy and the swirl strength from being transported to the turbine outlet at the diffusor part.

(3) As for the experimental results, after applying the new draft tube, despite the slight efficiency drop at the high water net head side, the model turbine efficiency performance at the low net head side is significantly improved by 10% at the part load, 20% at the design point, and 20% at the full load.

As for a draft tube design, this study successfully achieved the goal. But for research, some questions remained. Basing on the numerical results, despite a successful suppression on the swirl strength throughout all operating condition is predicted, the hydraulic loss at BEP still overrides that of the existing draft tube as a result. Therefore, further study including the improvements on numerical accuracy and detailed experimental visualization, as well as other possible factors or physic phenomena is expected.

Acknowledgments
This work was completed by the strong support of the project members in hydraulic research laboratory of Toshiba Corporation in Japan and Toshiba Hydro Power Co., Ltd in China. The authors would also like to acknowledge Kiyoshi Matsumoto, Takanori Nakamura, Yusuke Nakahara, and Norio Ohtake for helpful advices.

Product names mentioned herein may be trademarks of their respective companies.

References
[1] Ravindranath A. et al, “Mean Velocity and Decay Characteristics of the Near and Far Wake of a Compressor Rotor Blade of Moderate Loading”, J. Eng. Power 102(3), 535-547, 1980
[2] Gallus, H. E. et al, “Three Dimensional Separated Flow Field in the Endwall Region of an Annular Compressor Cascade in Presence of Rotor-Stator Interaction: Part 1”, J. Turbomach 112(4), 669-678, 1990
[3] Koutnik, J. et al, “Overload Surge Event in a Pumped-Storgae Power Plant”, 23rd IAHR Symp. On Hydraulic Machinaery & Systems, Yokohama Japan, 2006
[4] Chen, C. et al, “One-diemnsional Analysis of Full Load Draft Tube Surge”, ASME J. Fluids Eng., 130, 2008
[5] Alligné S. et al, “Prediction of a Francis Turbine Prototype Full Load Instability from Investigatations on the Reduced Scale Model”, 25th IAHR Symp. On Hydraulic Machinaery & Systems, Timisoara Romania, 2010
[6] Shingai, K. et al, “Long-Period Pressure Pulsation Estimated in Numerical Simulations for Excessive Flow Rate Condition of Francis Turbine”, Trans. ASME, J. Fluids Eng., 136, 2014
[7] Krappel, T., et al, “Investigation of Francis Turbine Part Load Instabilities using Flow
Simulations with a Hybrid RANS-LES Turbulence Model”, 27th IAHR Symp. On Hydraulic Machinery & Systems, Montreal Canada, 2014

[8] Magnoli, M. V. et al, “Influence of Hydraulic Design on Stability and on Pressure Pulsations in Francis Turbines at Overload, Part Load and Deep Part Load based on Numerical Simulations and Experimental Model Test Results”, 27th IAHR Symp. On Hydraulic Machinery & Systems, Montreal Canada, 2014

[9] Tamura, Y. et al, “Experimental and Numerical Study on Expansion of Steady Operating Range of Low Specific Speed Francis Turbine”, 13th Asian International Conference on Fluid Machinery, Tokyo Japan, 2015

[10] Feng, J. J. et al, “Investigation on pressure fluctuation in a Francis turbine with improvement measures”, 27th IAHR Symp. On Hydraulic Machinery & Systems, Montreal Canada, 2014

[11] Francke, Håkon Hjort, “Increasing Hydro Turbine Operation Range and Efficiencies Using Water Injection in Draft Tubes”, Doctoral thesis, Doktoravhandlinger ved NTNU, 1503-8181, 2010

[12] Marjavaara, B. D., “CFD Driven Optimization of Hydraulic Turbine Draft Tubes using Surrogate Models”, Doctoral thesis, Luleå University of Technology, 2006

[13] Menter, F. R., “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications” J. AIAA, Vol. 32 No.8, 1994

[14] Johnson, D. A et al, “Mathematically Simple Turbulence Closure Model for Attached and Separated Turbulence Boundary Layers”, J. AIAA, Vol. 23, No.11, 1985

[15] Bardina, J.E., Huang, P.G., and Coakley, T. J., “Turbulence Modeling, Validation, Testing, and Development” NASA Technical Memorandum 110446, April 1997

[16] Peter, R. et al, “Application of Multiblock Grid Generations with Automatic Zoning”, Numerical Grid Generation in Computational Fluid Dynamics and Related Fields, 1994