Proceedings of:
Current Research in Hydropower Technologies, CRHT X
Kathmandu University, 2020
CRHTX-05

Design optimization of Francis turbine draft tube

I Herland[1*]
[1]*Waterpower Laboratory, Department of Energy and Process Engineering, Norwegian University of Science and Technology, Alfred Getz Vei 4, Trondheim, Norway*

Abstract. This paper is based on ongoing master thesis work at NTNU with the topic: Design optimization of Francis turbine draft tube. The work is part of on-going research activities in the Waterpower Laboratory, Department of Energy and Process Engineering, NTNU and is done with collaborate research under H2020-HydroFlex-WP3 project. The focus is optimization of draft tube under variable-speed operating condition. Newly optimized runner design will be used to improve existing draft tube design. The object of the project is design and optimization of a high head Francis turbine draft tube for variable-speed turbine operation. The focus area is steady state operating conditions of the turbine. Ansys CFX is used for CFD calculations. Optimization results are not jet provided. The paper focus on motivation for doing design optimization, theoretical description of draft tube performance and description of the optimization method.

Keyword: Variable Speed Hydropower, Optimization, Draft tube, Francis turbines, HydroFlex

Introduction

Hydropower is the primary source of electricity in Norway, and it has enabled the domestic power industry for over 100 years [1]. With its advantages of low emissions and reliable supply, it is expected to play an important key role in the future energy mix. Predictions show that hydropower will grow with 78% world wide by 2050 [2]. Other renewable energy such as solar and wind energy are expected to grow as well, which contribute to variable power production due to variable solar and wind conditions and small storage capabilities. Fluctuations in the overall electricity production stresses hydropower plants as they are expected to dampen this effect. The stresses are related to that hydropower plants are operated far more roughly than before with extended periods of off-design operation [3].

Francis turbines are reaction turbine where operation condition is depended on available net head and the discharge. They are traditionally designed for synchronous speed operation, which leads to losses operating outside Best Efficiency Point (BEP), when net available head and/or discharge are different from the condition the turbine is designed for. The losses can by dived into incidence losses at the inlet of the runner and swirl losses at the outlet of the runner. By introducing variable speed as another degree of freedom, the best trade of between incidence and swirl losses can be found as a function of head and rotational speed of the runner. This will increase the overall efficiency of the runner, when operating outside BEP.

HydroFlex is a research project which aims to develop technology for more flexible operations of Francis hydropower plants in terms of large ramping rates, frequent start-stops, and possibilities to provide an extensive range of system services such as frequency and voltage regulations [4]. HydroFlex is organized by the Norwegian research centre for hydropower – HydroCen. This project has developed an optimized Francis runner design for operating at variable speed. This design is utilized further for draft tube optimization described in this paper.

Theoretical background

1.1. The draft tubes role in a hydropower plant

The draft tube's main task is to convert kinetic energy at the outlet of the runner into pressure energy at the outlet of the draft tube. Additional advantages of the draft tube are that it enables placing the runner above the tail water without losing head, as well as directs the water flow into the tail water. The draft tube increases the overall efficiency of the hydropower plant, which can be explained by Figure 1.

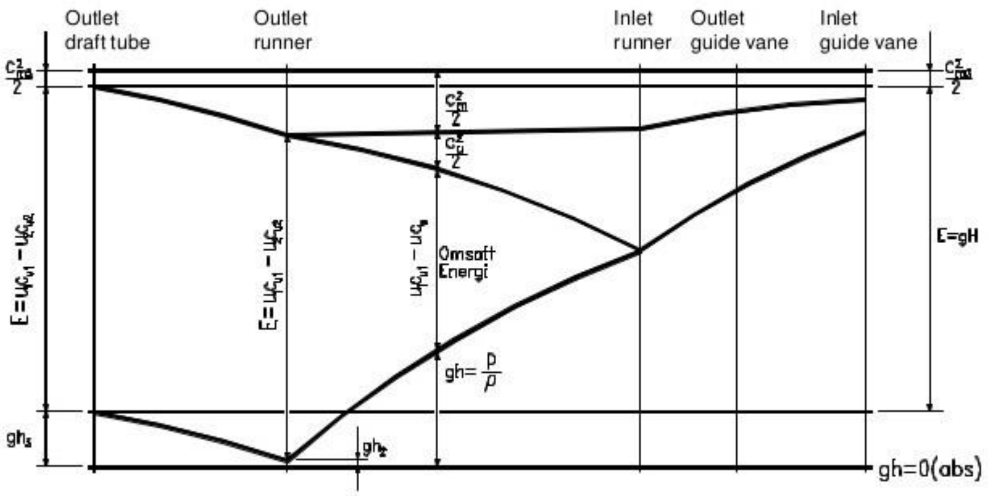

Figure 1. Specific energy conservation in a Francis turbine [1].

In a Francis turbine, both pressure energy and kinetic energy is transferred from the water into rotational energy of the runner. The remaining kinetic energy in the water at the outlet of the runner is considered as losses. By introducing a draft tube, a conversion of kinetic energy in the water at the runner outlet into pressure energy at the draft tube outlet is possible. The pressure at the outlet of the draft tube is bounded by the conditions at the tail water. Because of this, pressure recovery throughout the draft tube decreases the pressure at the outlet of the runner. By this, the total energy converted to the rotational energy of the runner will increase.

1.2. Measuring performance of a draft tube

The performance of the draft tube can be measured several ways. The pressure recovery factor is a dimensionless quantity, reflecting how much the static pressure has increased through the draft tube:

$$C_p = \frac{\frac{1}{A_{outlet}}\int P\, dA_{outlet} - \frac{1}{A_{inlet}}\int P\, dA_{inlet}}{\frac{1}{2}\rho\left(\frac{Q_{inlet}}{A_{inlet}}\right)^2} \qquad \textbf{(1)}$$

Where P is the static pressure, A is area, ρ is density and Q is volume flow. A high pressure recovery factor describes a good draft tube design from a hydrodynamic perspective. Draft tube performance can also be described with the energy loss coefficient, which reflect losses in stagnation pressure throughout the draft tube:

$$\xi = \frac{\frac{1}{A_{inlet}}\int P_t \, dA_{inlet} - \frac{1}{A_{outlet}}\int P_t \, dA_{outlet}}{\frac{1}{2}\rho\left(\frac{Q_{inlet}}{A_{inlet}}\right)^2} \quad \textbf{(2)}$$

Where P_t is the stagnation pressure. A low energy loss coefficient indicates low energy losses in the draft tube and good hydrodynamic performance. For experimental analysis, the pressure recovery factor is rather used than the energy loss coefficient, because the energy loss coefficient requires to know the entire velocity profile over the inlet and the outlet of the draft tube. However, for CFD analysis, both the pressure recovery factor and the energy loss coefficient are rapidly used for measure hydrodynamic performance [5,6,7]. A third parameter describing the draft tube empathized in this paper, is the total volume of the draft tube. Construction cost can be assumed to increase with increased volume of draft tube, and by this it will be benefitable to keep the total volume of the draft tube sufficiently small. Lastly, keeping track of the minimum value of the pressure can be an indicator about eventually cavitation, which should be avoided as it can cause damages.

1.3. Flow in a draft tube

The flow in the draft tube is complex due to factors such as unsteadiness, turbulence, separation, curvature streamline, secondary flow, swirl, and vortex breakdown [8]. Francis turbine designs give approximately no swirl of the water leaving the runner when it is operating at BEP [9]. This means that the water will enter the draft tube in the axial direction when the turbine is operating at design condition. However, when the turbine is operating outside design condition, the water will leave the runner with a swirl. For part load operation, the swirl will be in the same direction as the runner. In contrast, operating at full load gives rise to a swirl of water in the opposite direction of the runner. This can be seen from the direction of the vector velocities at the runner outlet in Figure 2.

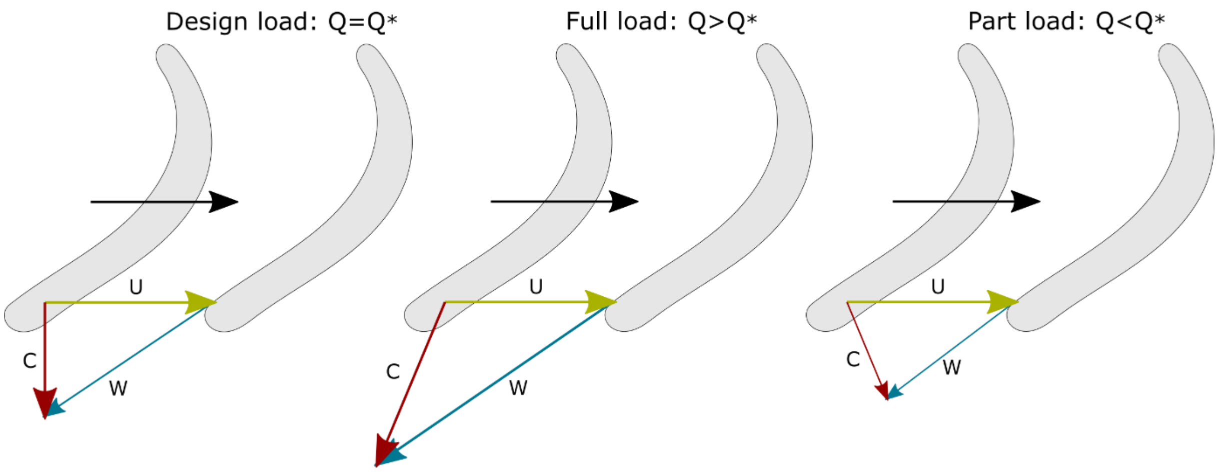

Figure 2. Velocity triangles at the outlet of the runner for different loads. c is the actual velocity of the water, w is the velocity of the water relative the runner blades and u is the velocity of the blades at the runner outlet.

The swirl of the water can give rise to a rotating vortex rope with a low pressure region inside. This again can lead to pressure pulsation in the hydropower plant as well as cavitation if the pressure decreases lower than the vapour pressure.

The overall geometry of a draft tube is a diffuser, where the cross section is gradually increasing to reduce velocity and increase pressure. If the walls of the draft tube are diverging too much, flow separation from the wall will occur, which will lead to secondary flow and pressure losses. The elbow draft tube is the todays most common design of draft tubes [8]. They consist of a diverging first part, the cone, a bent second part, the elbow, and a diverging last part, the exit diffuser. The bend or the elbow causes pressure losses, and flow separation typically occur around the inner surface downstream the bend. However, this elbow draft tube has much lower civil cost compared to a straight diffuser, making this design the preferred one.

Optimization method

Figure 3 shows the overall flow chart of the optimization method. Three different operation points where chosen for doing three separately optimization processes.

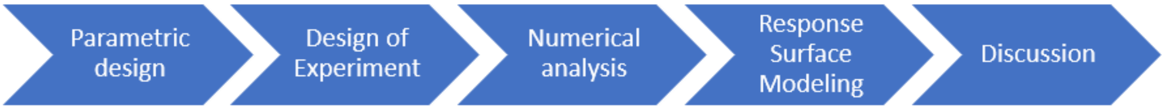

Figure 3. Flow chart of steps in the optimization process.

1.4. Operation points

The operation points chosen are based on a previously variable speed runner design optimization preformed by Igor Iliev as part of the HydroFlex Project. The operation points correspond to one part load(PL) , one best efficiency point(BEP) and one full load(FL) condition, and are summarized in Table 1.

Table 1. *Operation points for optimization.*

Type	$n\ [rpm]$	$Q\ [m^3/s]$	$\alpha\ [deg]$
PL	338.40	0.10577	5.434
BEP	348.43	0.19066	10.000
FL	348.43	0.22477	12.076

1.5. Parameterization

The draft tube geometry is described by several parameters illustrated in Figure 4. The geometry of the runner inlet is kept fixed to always match the runner outlet. The draft tube inlet consists of a small rotating wall, representing the runner hub and an outer surface with a small inclination where the water is entering from the runner outlet. The diameter of the cross sections in the cone increase linearly. In the elbow, the cross section geometry is changed from circular at the inlet to rectangular with bent corners at the outlet. The centreline of the bend is constructed with a Bezier curve with control points in P_1, P_2 and P_3. l is increased linearly from 0 to l_3 through the bend. d is adjusted so that the cross section area is following the graph in Figure 5. By this, the cross section area of the bend firstly increase before it is contracting a bit at the end. The mild diffuser effect would contribute to less flow separation as the flow more easily will follow the curvature of the wall [13]. The cross section area of the exit diffuser is created by letting the height, width and radius of the corners change linearly from the outlet of the elbow (d_3, $l_3 + d_3$, $d_3/2$) to the outlet of the draft tube (l_4, l_5, r_1). Ansys SpaceClaim is used to construct the draft tube geometry with a python script. The draft tube is created by constructing cross sections perpendicular to the bulk flow direction and lofting a volume between the surfaces.

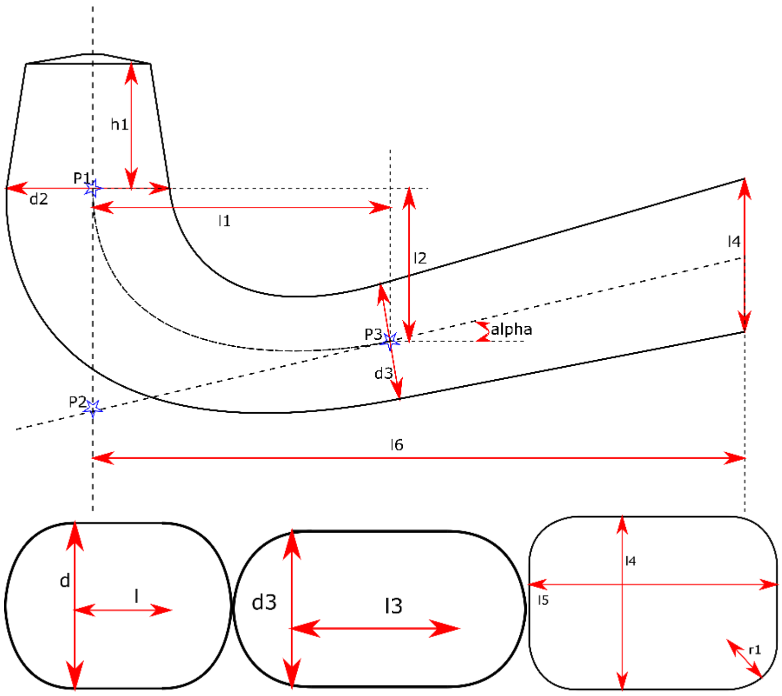

Figure 4. *Design variables of the draft tube and control points for the center curve through the bend section, illustrated with a sideview (upper) and cross sections perpendicular to the bulk flow motion inside the bend (bottom left), at the end of the bend (bottom middle) and the outlet of the draft tube (bottom right).*

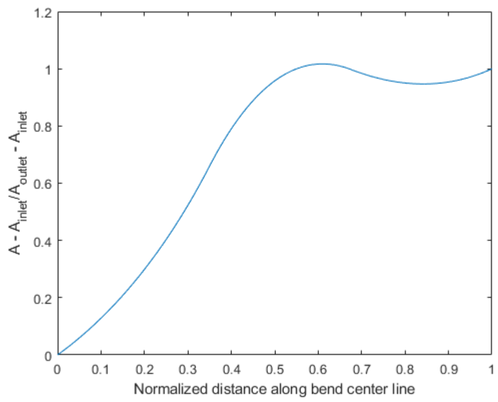

Figure 5. *Normalized cross section area relationship as a function of normalized distance in the bend.* A_{inlet} *is the cross section area at the inlet of the bend and* A_{outlet} *is the cross section area at the outlet of the bend.*

1.6. Design Space

Table 2 summarize the n = 11 design parameters used for the optimization. The center values are based on measurements done at a prototype reconstructing of the draft tube in Tokke Kraftverk, performed at NTNU during the Francis-99 Workshop. The design space is chosen by letting the design parameters vary ± 30% from their center value.

Table 2. Design parameters and design space

Design parameter: x_i	*Central value:* x_i^c	Lower limit: x_i^l	Upper limit: x_i^u
d_2	$404\ mm$	$283\ mm$	$525\ mm$
h_1	$477\ mm$	$334\ mm$	$620\ mm$
l_1	$1459\ mm$	$1021\ mm$	$1897\ mm$
l_2	$761\ mm$	$533\ mm$	$989\ mm$
α	$15.0°$	$10.5°$	$19.5°$
l_3	$277\ mm$	$194\ mm$	$360\ mm$
d_3	$308\ mm$	$216\ mm$	$400\ mm$
l_4	$639\ mm$	$447\ mm$	$831\ mm$
l_5	$588\ mm$	$412\ mm$	$764\ mm$
l_6	$4374\ mm$	$3062\ mm$	$5686\ mm$
r_1	$160\ mm$	$112\ mm$	$208\ mm$

1.7. Objectives

The objectives of an optimization process are different quantities we either want to maximize, minimize or control by the optimized design. The objectives for this optimization process are the pressure recovery factor, the energy loss coefficient and the volume of the draft tube. An optimization process with more than one objective is called a multi-objective optimization process. In mathematical terms, the optimization process can be expressed as:

$$\begin{aligned} &Maximize\ C_p(x) \\ &Minimize\ \xi(x) \\ &Minimize\ V(x) \\ &x \subseteq \mathbb{R}^n, x \in x\ subject\ to\ x_i^l \leq x_i \leq x_i^u, i = 1,2,3,\dots,n \end{aligned} \quad \textbf{(3)}$$

1.8. Design of Experiment

Design of Experiment (DOE) are methods used for populating the design space in samples, meaning selecting different combinations of the design parameters within the design space. The samples are further evaluated numerically with CFD analysis and their corresponding objective values are found. These results are further used to create models of the objectives as functions of the entire design space. An efficient DOE method obtains the maximum amount of information with minimum number of design samples, as this will give good accuracy of the results vs. CPU time required for obtaining them.

Box-Behnken designs (BBD) is a DOE method developed by George E. P. Box and Donald Behnken in 1960 [10]. It is a three level method, meaning that it uses combinations of the design parameters from three levels; an upper, a lower and a center value. Compared to the DOE method, three level full factorial design (TLFFD) which is combining all possible combinations of design parameters from three levels, BBD is only using a restricted number of combinations and is avoiding extreme conditions. Figure 6 shows BBD compared with TLFFD sampling example for three design parameters.

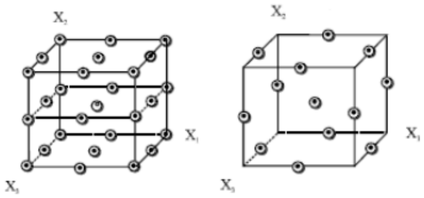

Figure 6. *Three Layer Full Factorial design (left) compared with Box-Behnken design (right) with three design parameters* [14].

For BBD, the number of samples are given by $m = 2n(n-1) + CP$, where n is number of design parameters and CP is the number of center points calculations (sometimes more than one to reduce the variance). The optimization problem discussed in this paper got n = 11 design parameters, which will result in m = 220 + CP samples. By comparison, TLFFD results in $m = 3^n = 177147$ samples, which will require significantly larger CPU time. Another advantage with BBD is that it has been constructed for use together with second order response surface models (RSM), which is the model chosen for objective modeling, described in section 3.7.

1.9. Numerical analysis

Ansys CFX is used for CFD analysis of the samples. It is a closed source software, meaning that the source code is not provided, but a detailed description of functionalities can be found in the Ansys CFX user guide [11]. For the optimization problem, RANS equations are solved with the SST turbulence model. To evaluate numerical error the GCI method [12] will be used. Validation of the numerical results would not be possible, as experimental data does not exist for the new draft tube design provided in the optimization process. However, this can be suggested as further work if a prototype of the optimized draft tube design is built.

To ensure that the optimization process turns out to be as efficient as possible, several test simulations will be done in advance. It is desirable to run the optimization simulation without the runner, as this will make the computations less expensive. This require an assumption about that the inlet condition of the draft tube must be approximately unchanged for different draft tube design. To test if this assumption hold, simulations of three different draft tube design together with runner and guide vane will be done. The inlet profile of the draft tube will be compared, and if the results shows negligible differences, the optimization process will be performed with the draft tube only. Additionally, simulations will be performed to test how the objectives eventually vary for transient vs. steady state condition. It is desirable to do steady state simulations in the optimization process as this is less expensive, even though complex flow in a real draft tube is transient by nature. The test evaluation described will be done for the three different operation points separately.

For CFD calculations of guide vane, runner and draft tube together, mas flow and unit velocity vectors will be given as boundary conditions at the guide vane inlet and average static pressure equal zero will be given as an outlet condition at the draft tube outlet. For eventually only draft tube simulations in the optimization process, the velocity magnitude and direction will be exported from the test simulations and given as an inlet condition at the draft tube inlet. The boundary condition at the draft tube outlet will be average static pressure equal zero here as well.

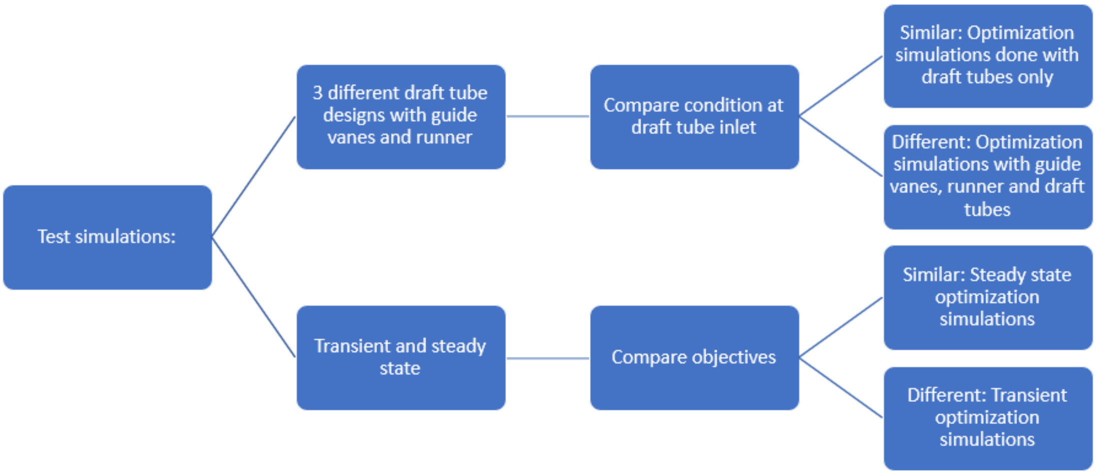

Figure 7. Flow chart of test simulations and further decisions.

1.10. Response surface modeling

Second order response surface models (RSM) will be built based on the CFD results to create mathematical models of the objects based on the design variables. RSM are performed by approximate an unknown objective function $\boldsymbol{y}(\boldsymbol{x})$ with a second order polynomial:

$$\hat{y}(x) = \beta_0 + \sum_{i=1}^{n} \beta_i x_i + \sum_{i=1}^{n} \beta_{ii} x_i^2 + \sum_{i<j}^{n} \sum_{j=2}^{n} \beta_{ij} x_i x_j \quad \textbf{(4)}$$

Where $\hat{y}$ is CFD objective results, x_i are linear terms, x_i^2 are quadratic terms, $x_i x_j$ are mixed terms, and β are regression factors which needs to be determined. The minimum number of samples, p, needed to decide the unknown regression factors are depended on the number of design parameters, n, and given by the relation $p = \frac{(n+1)(n+2)}{2} = 66$, when $n = 11$. The difference of the RSM approximation and the real objective function are represented with a random error, ε:

$$y(\boldsymbol{x}) = \hat{y}(\boldsymbol{x}) + \varepsilon \quad \textbf{(5)}$$

Since the system is overdetermined, $m = 220 \geq p = 66$, the regression coefficients are estimated by a least square minimization. In matrix notation, this can be expressed as:

$$\boldsymbol{y} = \boldsymbol{X\beta} + \boldsymbol{\varepsilon} \quad \textbf{(6)}$$
$$\boldsymbol{\beta} = (\boldsymbol{X^T X})^{-1} \boldsymbol{X^T y} \quad \textbf{(7)}$$

Where $\boldsymbol{\beta}$ is a $p \times 1$ column vector of the regression coefficients and $\boldsymbol{X}$ is a $m \times p$ data matrix containing the m rows of samples and p columns of the linear, quadratic and mixed terms.

Two different quantities are common for checking the accuracy of the RSM, called checking the goodness-of-fit. The first is the root mean squared error σ_e, defined as:

$$\sigma_e = \sqrt{\sum_{i=1}^{m} (\boldsymbol{e}^{(i)})^2 / \boldsymbol{m}} \quad , \quad \boldsymbol{e}^{(i)} = \left\| \frac{\hat{\boldsymbol{y}}^{(i)} - \boldsymbol{y}^{(i)}}{\boldsymbol{y}^{(i)}} \right\| \quad \textbf{(8)}$$

The second is the adjusted R-squared coefficient of determination R^2_{adj}, defined as:

$$R^2_{adj} = \mathbf{1} - \frac{\sum_{i=1}^{m}(y^{(i)} - \hat{y}^{(i)})^2}{\sum_{i=1}^{m}(y^{(i)} - \bar{y}^{(i)})^2} \cdot \frac{m-1}{m-p} \quad \textbf{(9)}$$

where $\bar{y}$ is the mean of the simulated output parameters. An ideal fit will result in $\sigma_e = 0$ and $R^2_{adj} = 1$.

1.11. Discussion of the results

The optimal draft tube design would further be discussed based on the response surface model. It is important to emphasise that the goal of the optimization process not will be to obtain an accurate result of the most optimal design, but to capture the most important trends of how different design parameters influence the overall draft tube performance.

The optimal design for each objective individually can be found by searching for the design parameters corresponding to either the maximum or minimum value of the response surface models. Pareto fronts can be used to investigate possibilities for obtaining an optimal design for more objectives at the same time. A sensitivity analysis can also be performed to compare how much different design parameters influence the objectives.

It is expected that a more diverging cross section area of the draft tube in total, will lead to higher pressure recovery factor. However, if the walls are diverging to much, the present of flow separation would lead to pressure losses and energy losses. It will be interesting to investigate the difference between the different operation points. It is expected that PL and FL can handle more diverging walls in the draft tube cone without having flow separations at the wall, compared to BEP, as rotation of the water dampen eventually flow separation.

Conclusion

This paper summarize theory about draft tube performance and characterize $\boldsymbol{C_p}$, ξ and V as main objectives of interest. How to do draft tube design optimization with CFD are further described, where three operation points (PL, BEP and FL) are considered separately. The main ideas behind BBD and RSM are explained as well as how the results can be discussed to find an optimal design. Since the optimization process are based on investigating new design, a verification of the CFD results will not be possible unless prototypes are built and tested. However, the mesh quality can be evaluated with the GCI method and the accuracy of the RSM should be checked by the goodness-of-fit to ensue reliable results.

References

[1] H. Brekke: Introduction to hydraulic machinery, 2000, Compendium; NTNU Waterpower laboratory.

[2] DNV GL: DNV GL Energy transition outlook 2019, 2019, *DNV GL AS*

[3] L. E. Jones 2017 Renewable energy integration: Practical management of variability, uncertainty and flexibility in power grids *Elevier*, *Academic Press* **2** p 530, ISBN: 9780128095928

[4] HydroCen, 2018. Design of a high-head Francis turbine for variable speed operations. https://www.ntnu.edu/hydrocen/high-head-francis-turbine-for-variable-speed-operation. [Online; accessed 15-December 2019].

[5] C. N. Mun, D. C. Ba, X. j. Yue and M. I. Kim, "Multi-Objective Optimization of Draft Tube in Francis Turbine Using DOE, RBF and NSGA-II", Preprints.org; 2017. DOI: 10.20944/preprints201704.0148.v1.

[6] M. C. Nam, "Design optimization of hydraulic turbine draft tube based on CFD and DOE method", *et al* 2018 *IOP Conf. Ser.: Earth Environ. Sci.* **136** 012019

[7] M. H. Shojaeefard, A. Mirzai and A. Babaei, "Shape optimization of draft tubes for Agnew microhydro turbines," *Energy Conversion and Management* 2014, Vol 79, pp 681-689.

[8] B. D. Marjavaara, "CFD Driven Optimization of Hydraulic Turbine Draft Tubes using Surrogate Models", 2006, Doctoral Thesis Luleå University of Technology.

[9] S. L. Dixon and C.A. Hall, "Fluid Mechanics and Thermodynamics of Turbomachinery", 2014, Elsevier Inc.

[10] G. E. P. Box and D. Behnken, "Some new three level designs for the study of quantitative variables", Technometrics, Volume 2, pages 455–475, 1960.

[11] ANSYS, Inc. "ANSYS CFX-Solver Theory Guide", 2016, Release 17.0.

[12] I. B. Celik, U. Ghia, P. J. Roache and C. J. Freitas, "Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Application," *Journal of Fluids Engineering* 2008, Volume 130, doi: 10.1115/1.2960953

[13] E. Mosonyi, "Water Power Development Low Head Power Plants", 1987, Third ed. Akademiai Kiado, Budapest. Pp. 647-653, 669-679.

[14] E. G. F. Nunez, R. V. Veliz, B. L. V. da Costa, A. G. de Rezende and A. Tonso, "Using Statistical Tools for Improving Bioprocesses", *Asian Journal of Biotechnology* 2013, Volume 5, 1-20.