Fully automated multidisciplinary design optimization of a variable speed turbine

Erik Tengs¹, Flora Charrassier¹, Maria Rolstad Jordal¹, Igor Iliev²
¹EDR&Medeso AS, Lysaker Torg 45, 1366 Lysaker, Norway
²Norwegian University of Science and Technology, The Waterpower Laboratory, 7491 Trondheim, Norway
E-mail: erik.tengs@edrmedeso.com

Abstract. The future electricity market will have large contributions from renewable energy sources such as solar and wind. The intermittent nature of these energy sources creates a need for highly flexible operation of hydropower stations and changes the way we use hydraulic turbines to more off-design operation and more start-stop cycles. These changes challenge the structural integrity of the turbines in a way not seen before, and the next generation of hydro turbines will therefore have to be designed differently to meet this challenge. The goal of this article is to develop a framework for variable-speed Francis turbine design.

A fully automated multi-disciplinary design optimization procedure has been developed. As off-design operation is assumed, the runner do not only have to be optimized from a hydraulic point of view, the structural integrity is equally important. The design optimization is therefore based on a blending function of the hydraulic efficiency and the harmonic stress levels at a series of operating conditions. This is to ensure that the turbine is less prone to fatigue, even at off-design operation.

The process is fully automated, with no need for human interaction. A MATLAB design code is producing the raw design, every design is then meshed and tested at different operating points in a CFD solver. The pressure field from the fluid analysis is mapped onto the structure and evaluated in an acoustic-harmonic analysis to assess the fluctuating stresses in the turbine blades. A global optimization loop consisting of 15 design parameters is driving the process based on an overall optimization function.

The work has been performed as part of the HydroFlex project, and has received funding from the European Union’s Horizon 2020 research and innovation programme under grant agreement No 764011.

1. Introduction

The current transition of the European energy systems towards a sustainable and renewable production indicates that intermittent renewable energy sources, such as wind and solar, will be key components in the future energy mix [1]. This increases the need for technologies which can balance the system by providing flexibility. Hydropower is the largest energy storage technology currently available. In fact, more than 95% of all energy storage capacity in the world is in pumped hydro [2]. Additionally, hydropower can provide flexibility to the energy market since hydropower plants are able to start and stop production independent of the weather, unlike wind and solar. Hydropower stations will be operated in a highly flexible way, and the hydraulic turbines will be used at off-design and have significantly more start-stop cycles.
However, current turbine designs have certain limitations when it comes to frequent variability. When these turbines try to accommodate the energy fluctuations in today’s electricity market, they experience increased strain and fatigue [3, 4]. This results in a reduction of operating lifetime. Hence, the development of technological solutions that accommodate frequent variability, such as variable speed operation, is key for the future electricity market and for improving power grid stability.

The goal of this paper is to present a framework for multidisciplinary optimization of variable-speed Francis turbine runner design, where off-design operation is assumed. It is acknowledged that optimal hydraulic design and optimal structural integrity are conflicting interests, and any optimization procedures should therefore include criteria from both physical domains. In this paper, no direct optimization have been performed, instead, the parameter space have been suitably sampled, to indicate the importance of the different parameters on the hydraulic and structural performance respectively. The main result from this paper is the presented framework, not a finished turbine design. Any final designs should be verified using high-fidelity simulations both on the hydraulic and structural side.

The work has been performed as part of the HydroFlex project [5].

2. Parametric design tool
A parametric design tool will be built using the design optimization tool ANSYS OptiSLang. This software will link MATLAB to the parametric modeling capabilities of ANSYS Workbench. A turbine design is created using a parameterized MATLAB design code, and based on this, a geometry and numerical mesh will be created for both the fluid and structural domain. The geometry will be passed on to the simulation loop, where a number of simulations at different operating points will be performed. A one-way fluid-structure interaction (FSI) will be performed by mapping the oscillating pressure forces from a Computational Fluid Dynamics (CFD) analysis onto the structure to perform harmonic analyses. OptiSLang will collect the different responses, and a new design will be dispatched. Figure 1 illustrates the procedure.

Figure 1: Conceptual drawing of optimization framework

2.1. Turbine Design
A design code is central to a parametric, automatized optimization procedure [6, 7]. This code needs to fulfill certain requirements, specifically, it has to be parametric, meaning it has to produce designs without human interaction, and it have to be able to produce a significant range of designs.

In this work, a MATLAB based design code is used. The code is developed at the Norwegian University of Science and Technology (NTNU), as part of the HydroFlex research project. The MATLAB code combines the classical design theory for turbomachinery [8] together with
techniques for parametric geometry definition. The goal of the HydroFlex project is to create a replacement runner for the Francis-99 reference case [9]. The code should therefore provide flexible geometry generation options with a high variation within the constrained design space of the existing Francis-99 turbine pit. Please refer to [10, 11] for detailed information regarding the design tool and the theory behind.

15 geometric parameters are used to define the runner geometry. The meridional view of the runner is defined using Bezier curves, while the meridional velocities are calculated using potential flow theory. The final geometry of the blade is defined using flexible blade angle distribution functions again defined as Bezier curves. A brief description of the respective parameters is given here:

- $M_{1,2,3,4}$: Controls the shape of the hub curve in the meridional view. It is a Bezier curve with 4 controls. The shroud curve is fixed and same as for the Francis-99 original runner.
- $T_{1,2}$: Inputs for a Bezier curve that control the location and shape of the trailing edge in the meridional view of the runner.
- $\Phi_{SL1}$: Controls the wrapping angle of the complete blade
- $\Phi, \Phi_{TE}$: Leaning of leading and trailing edge respectively
- $\Delta \beta_{1,2,3}$: Parameters controlling the outlet blade angle
- $H$: Deviation from design Head
- $Q$: Deviation from design Flow
- $h_1$: This parameter represents the maximum thickness of the blade.

Figure 2 illustrates the design constraints in the turbine design code. All parameters are normalized to a range of -1 to 1, and managed to produce designs in the complete range. The absolute range, and reasoning behind, is discussed in [11].

Figure 2: Different variable combinations and its effect on the blade design.
2.2. Mesh Generation

After the design have been created, the fluid and structural mesh is produced. The output from
the MATLAB turbine design code is a set of text files with coordinates describing the hub,
shroud, blade hydraulic surfaces and the constant outer geometry of the runner. Figure 3 shows
the general flow of files.

![Mesh generation process](image)

Figure 3: Mesh generation process

2.2.1. Fluid Domain  ANSYS TurboGrid, a tool for meshing fluid domains in turbo-machinery
is used to create the CFD mesh. The meshing is scripted with constant mesh refinement settings,
such that all the tested designs have similar mesh refinement. TurboGrid creates the mesh for
one flow passage, with node matching mesh on the periodic interfaces. The number of nodes in
the full runner with 17 blades was about 850 000.

2.2.2. Structural Domain  The 3D modeling software ANSYS SpaceClaim is used to generate
the structural geometry. Text files describing the geometry is used to create a complete 360°
turbine runner, before being cut into one section to speed up the simulation process. Figure 4
shows the transition from text files to complete structural geometry. Note as well the acoustic
domain, used to model the fluid. This ensures that the added mass effect is properly modeled
and any impact of an acoustic pressure component is captured.

![Curves read into SpaceClaim](image)

Figure 4: Curves read into SpaceClaim (a) used to automatically create a full (b) and cyclic (c)
structural turbine design and acoustic elements
The structural turbine design is then imported into ANSYS Mechanical for meshing. As in the fluid domain, the mesh settings was constant to ensure similar mesh refinement in all simulations, resulting in a mesh of approximately 100 000 nodes per flow passage.

2.3. Numerical Simulations

After generation, the numerical meshes are imported into the simulation loop, ref. figure 1. In the simulation loop two different simulations are performed. A CFD analysis is performed in the fluid domain, and a Finite Element Analysis (FEA) is performed in the structural domain. The key output from the CFD analysis is the hydraulic efficiency, as well as the fluid pressure. The oscillating fluid pressure is loaded onto the structure, and the structural stresses are calculated. The following sections will cover the details of the different simulations.

2.3.1. Computation Fluid Dynamics

CFD is becoming the standard tool for design of hydraulic turbines [12, 13]. In this work, because we are assessing the stresses in the structure due to oscillating pressures, transient simulations are performed. The subsequent structural analyses are performed in the frequency domain (Harmonic Analyses), and the pressure load from the CFD analyses will therefore have to be expressed in the same way. This is done using a Fourier series:

\[ p = a_0 + \sum_n a_n \cos(n\omega t) + b_n \sin(n\omega t) \]  \hspace{1cm} (1)

The export of pressure Fourier coefficients have to be done automatically to be possible to use in an optimization workflow. The solution to this is to run simulations using phase-shifted periodic boundaries. This means that only a small section of the runner is simulated and the periodic boundaries are prescribed using a Fourier series in time and space. Using this method, which is native in ANSYS CFX, the Fourier coefficients will be generated automatically, ready to use in the structural simulations.

The guide vanes was meshed with different openings in advance, and loaded into the simulations along with the runner, see 5a. Otherwise, the setup followed many of the recommendations in [14], using total pressure/pressure boundary conditions. The direct output of the CFD analysis was the hydraulic efficiency, as well as the Fourier coefficients of the pressure field.

Figure 5: (a) Two passages used in the CFD analysis. (b) Structural mesh with pink surfaces indicating the mapped pressure from the CFD analysis, and blue surface indicating the fixed displacement
2.3.2. Finite Element Analysis  Static and Harmonic acoustic analysis is run to predict the sustained static and dynamic behavior of the turbine with added mass effect from the fluid. Cyclic symmetry is used to reduce simulation time implying the use of a mesh match control on symmetry faces. A non-cyclic pressure load is defined in Mechanical using ANSYS Parametric Design Language (APDL) and the previously exported Fourier coefficient from CFD analysis. The shaft connection surface is defined as fixed in the analysis, see figure 5b. Stresses extracted along the trailing edge are the output of the structural analysis.

2.4. Optimization loop
The optimization is done in optiSLang as a sensitivity study consisting of two nested loops. The parameters for the turbine design is set in the outer loop, where new designs are generated in the MATLAB design code. To evaluate the design, simulations at different operating points is performed. This is done in the inner loop. Selected outputs from the simulations are transferred to the outer loop to evaluate each design.

2.4.1. Operating points  To optimize the maximum hydraulic efficiency, only the result at Best Efficiency Point (BEP) is necessary. However, variable speed is the goal in this work and the turbine performance at off-design must therefore also be evaluated. Turbine operation is usually defined based on the flow rate, \( Q \), and the runner speed, \( rpm \). The selected operating points was chosen to be in the range \( \pm 20\% \) of the respective value at BEP. In this work, simulations at five predetermined operating points was done for each design. This corresponds to a full factorial and center-point sampling where the center point result is the BEP. See figure 6 for an illustration.

![Figure 6: Operation points used in CFD and FEA analysis, example efficiencies](image)

2.4.2. Optimization functions  The presented framework is capable of implementing a variety of different optimization procedures. In this work we have adopted a multi objective optimization approach with one optimization function from each physical domain being fed back into the global loop.

The creation of the two responses from the inner loop presents the user with a very important choice; How to obtain optimization functions that properly reflect the overall goal? A variable speed turbine will need a good efficiency in the complete operating range, as well as low stresses.
Looking at the efficiency, a simple mean of the results seems intuitive, but by evaluating only the mean, information about the max efficiency and the variability of the results is lost. More advanced functions such as standard deviations can be included. This is a user choice, and depends on the expected use of the turbine. For simplicity, the mean was chosen as the efficiency response in this work. A similar argument can be presented for the structural domain, where the stresses are evaluated. Here, a max function was used to extract the maximum stress values along the trailing edge of the runner blades.

2.4.3. Outer optimization loop In the outer loop, a sensitivity analysis was performed. This means that no direct optimization procedure (e.g. Simplex [15]) was used. The Latin Hypercube [16] sampling procedure was used to properly sample the solution space.

3. Results

The goal of the present study was to create a functioning framework for optimization. Figure 7 shows the final setup as a flow chart. Several test runs have been performed on the presented workflow. In a typical run, ≈ 5% of the simulations failed to complete. Typical reasons for failure include failure during automatic geometry or mesh generation. Network connectivity to the computational cluster was also identified as a possible reason for failure. This will not affect the overall sensitivity analysis, other that creating a void design, and was deemed to be acceptable at this stage.

![Flowchart of the final setup](image.png)

For a optimization scheme to be of industrial interest, the simulation time needs to be as small as possible. Effort was put into reducing this as much as possible, and in the current setup, each operating point with CFD and FEA analysis was finished in well below one hour (on a 40 core computational cluster). This means that each design can be evaluated in a couple of hours, and large scale testing is feasible.
3.1. Comparison with previous work
As mentioned in section 2.1, the turbine design code used in this work is taken from \[10\], co-authored by the present authors. A sensitivity analysis of the CFD response of the different designs was performed. For detailed information refer to the referenced paper. Figure 8 shows the sensitivity of the hydraulic efficiencies at best efficiency (BEP) based on that work. Several interesting conclusions can be made based on this figure, notably that a few variables account for most of the variability in the efficiency.

![Figure 8: Sensitivity study of the efficiency at BEP](image)

Comparing this to preliminary results from the current framework shows very similar results. Figure 9 shows the Coefficient of Prognosis (CoP) matrix. In simple terms, this shows how much of the variability in the results are explained by the respective free variables. Note that not all variables are shown, as they have small effects on the overall results. We see that the dominating variables from the reference is also identified in this work, with one discrepancy being $\Phi$, which seems to be overvalued in this work.

![Figure 9: CoP at BEP](image)

The present study differs from the reference in a couple of important ways. Firstly, the CFD simulations are transient, which can explain some of the discrepancies in the efficiency, and more importantly, the structural response is included. In the referenced work (figure 8), one might come to the conclusion that certain variables (e.g. $h_1$) can be disregarded due to
low variability. In the structural response however, \( h_1 \) is hugely important (figure 9). This has an obvious explanation, as \( h_1 \) controls the thickness of the blade, but is something that a pure CFD optimization would miss. The procedure presented here will therefore potentially illuminate some design constraints not known before.

4. Discussion and further work

It should be noted that the CFD and FEA analyses presented here are fairly coarse simulations, and numerical inaccuracies may occur. It has not been the goal to present a perfect numerical simulation, nor is experimental data available for validation. We are therefore focusing on the methodology and workflow itself, rather than exact designs, efficiencies and stress values.

Further work A natural next step is to update the CFD and FEA modules with more detailed analysis. In addition, the number of operating points per design may be increased to get a full overview of the performance of each design at variable speed. The number of optimization functions and the optimization goal may also be adjusted per user request. As the setup becomes more detailed, results from the sensitivity study may allow for setting certain design parameters constant and continue the optimization of the design with only those that make a difference to reduce the complexity and reduce simulation time. Finally, a full sensitivity analysis should be performed.

A way of evaluating the designs, when the full sensitivity analysis is done, is to use a Pareto-front. With two optimization functions as presented here, one can get an overview of the complete solution space by the use of a single graph. Figure 10 illustrates this. Here we can see some of the sampled designs, with the two optimization functions along the axes. The functions are scaled to the best result, to form a range of [0-1]. We see that the different design have vastly different performance, varying in the range of > 5% in efficiency, and close to 90% in the stress values. Please note that these are preliminary results.

The optimized design will be validated with laboratory tests in later stages of the HydroFlex project. These tests will be used to further improve the current simulation framework, as well as high-fidelity numerical simulations. It should be noted that it is only feasible to pre-calibrate
the simulation setup for a limited number of designs and operating points. Because of this, there is always a question of whether or not the numerical model provides good and converged results for all designs. To minimize the risk of including void results in the optimization loop, output parameters describing convergence will be included, to exclude such designs.

5. Conclusion
A fully automatized multidisciplinary design optimization tool for hydraulic turbine runners is presented. The tool is based on a MATLAB design code, and uses various ANSYS tools for simulation in both the fluid and structural domain. The oscillating pressure from a CFD analysis is mapped onto the structure, and the maximal stresses along the trailing edge is extracted. This, along with the hydraulic efficiencies are the optimization functions chosen. Several off-design operating points are tested, to improve performance at these conditions. Ideally, this can lead to a variable speed turbine design. The framework was very successful and showed great potential for future design.

Acknowledgement
This project has received funding from the European Union’s Horizon 2020 “Secure, Clean and Efficient Energy” programme, H2020-LCE-07-2016-2017, under grant agreement no 764011. Project: Increasing the value of hydropower through increased flexibility - HydroFlex [5].

References
[1] IEA world energy outlook 2019. https://www.iea.org/reports/world-energy-outlook-2019. Accessed: 2020-01-14.
[2] DOE global energy storage database. https://www.energystorageexchange.org. Accessed: 2020-01-13.
[3] Peter Dörfler, Mirjam Sick, and André Contu. Flow-induced pulsation and vibration in hydroelectric machinery: engineer’s guidebook for planning, design and troubleshooting. Springer Science & Business Media, 2012.
[4] U Seidel, C Mende, B Hüblner, W Weber, and A Otto. Dynamic loads in francis runners and their impact on fatigue life. In IOP conference series: earth and environmental science, volume 22, page 032054. IOP Publishing, 2014.
[5] Hydroflex. https://www.h2020hydroflex.eu/ Accessed: 2020-01-14.
[6] AE Lyutov, DV Chirkov, VA Skorospelov, PA Turuk, and SG Cherny. Coupled multipoint shape optimization of runner and draft tube of hydraulic turbines. Journal of Fluids Engineering, 137(11):111302, 2015.
[7] kazuyuki Nakamura and sadao Kurosawa. Design optimization of a high specific speed francis turbine using multi-objective genetic algorithm. International Journal of Fluid Machinery and Systems, 2(2):102–109, 2009.
[8] Hermod Brekke. Pumper & turbiner. Vannkraftlaboratoriet NTNU, 2003.
[9] francis-99 research project. https://www.ntnu.edu/nvks/francis-99 Accessed: 2020-01-13.
[10] Igor Iliiev. Hydraulic optimization of francis turbines for variable speed operation using surrogate modeling. Journal of Fluids Engineering, 2020 Accepted, not published.
[11] Igor Illiev. Francis turbines for variable speed operation. PhD thesis, Waterpower laboratory, NTNU, 2020. ISBN 978-82-326-4511-4.
[12] B Nennemann, TC Vu, and M Farhat. Cfd prediction of unsteady wicket gate-runner interaction in francis turbines: A new standard hydraulic design procedure. Technical report, 2005.
[13] P Drtina and M Sallaberger. Hydraulic turbines—basic principles and state-of-the-art computational fluid dynamics applications. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 213(1):85–102, 1999.
[14] Chirag Trivedi, Michel J Cervantes, and Ole Gunnar Dahlhaug. Numerical techniques applied to hydraulic turbines: A perspective review. Applied Mechanics Reviews, 68(1), 2016.
[15] John A Nelder and Roger Mead. A simplex method for function minimization. The computer journal, 7(4):308–313, 1965.
[16] Michael D McKay, Richard J Beckman, and William J Conover. Comparison of three methods for selecting values of input variables in the analysis of output from a computer code. Technometrics, 21(2):239–245, 1979.