DRAFT-TUBE INLET VELOCITY PROFILE OPTIMIZATION

Xin Lin Li[1], François Guibault[2], Christophe Devals[3]
[1]*Mech. Eng. Department, École Polytechnique de Montréal, xinlinli170@gmail.com*
[2]*Comput. Eng. Department, École Polytechnique de Montréal, francois.guibault@polymtl.ca*
[3]*Comput. Eng. Department, École Polytechnique de Montréal, christophe.devals@polymtl.ca*

Keywords: *Hydraulic turbine, Draft-tube, CFD, Optimization, Mesh Adaptive Direct Search*

This paper describes a methodology to formulate and solve an inlet velocity profile optimization problem to minimize hydraulic turbine draft-tube losses. The proposed approach is based on the Mesh Adaptive Direct Search (MADS) optimization algorithm coupled to an incompressible RANS CFD simulation, using OpenFOAM and the standard $k - \epsilon$ turbulence model. Sample results for the Porjus U9 draft-tube are presented. The results show that the energy loss factor was reduced by more than 60% in optimization cases compared with the best efficiency point found using a solid body rotation test. These optimization results can be used as a design reference for turbine designers working on rehabilitation projects of hydraulic power plants.

Introduction

Although global demand for renewable energy is growing steadily and hydropower plays a vital role in this growth, the number of dams built annually tends to decrease. This contradiction has prompted engineers to find ways to extract more energy from existing hydroelectric plants instead of building new ones. The rehabilitation of existing hydropower plants - thus increasing energy production while extending their life - is becoming increasingly important. This work finds its motivation in the needs of engineers for design tools adapted to the rehabilitation of the hydroelectric plants.
The hydraulic turbine is the central technological element involved in the conversion of hydraulic energy. Among the different types of hydraulic turbines, the Kaplan turbine is the most widely used axial turbine in the world. This type of turbine allows efficient hydropower generation in the case of high flow conditions and low head. The draft-tube is one of the most important components of axial turbines. It converts the dynamic pressure of the flow into static pressure by decelerating the flow before it returns into the downstream river. It accounts for 20% to 50% of the total energy that can be recovered from a low-head power plant [1]. The performance of the draft-tube depends on speed distribution at the inlet of the turbine and other factors such as cavitation, downstream water depth, turbine operating point, drag, detachments and secondary flows. All of these factors depend not only on the geometrical shape of the draft-tube, but also strongly on the design of the turbine runner.
In a project to rehabilitate a hydroelectric plant, the spiral casing and draft-tube are usually retained because they are part of the dam and are usually constructed of concrete. Some components such as the generator, the guide vanes and the runner are replaced. Therefore, installing a newly designed runner that better matches the existing draft-tube is the most practical and effective way to improve the overall efficiency of power generation for the entire turbine.
Traditionally, to get the best fit between runner and draft-tube, tests on several models were conducted to check runner designs. However, these model tests are very expensive, so that turbine designers can not explore the optimization space in a thorough and systematic way. The final design of the turbine runner is therefore generally a practical design rather than an optimal one.
Nowadays, with the rapid development of high-performance computing and high-fidelity CFD models, designers are able to obtain accurate low-cost predictions about draft-tube flow and predict the performance of the draft-tube without performing expensive tests. An improved design of the turbine runner, corresponding best to the existing draft-tube, can thus be obtained on the basis of these new techniques. Previous results have shown that the replacement of the turbine runner, achieved by modern technologies, while keeping other existing turbine structures, made it possible to increase the power output of a hydroelectric plant by a factor that could reach from 10 to 30% [2, 3]. However, current studies on the optimization of hydraulic turbines are still mainly focused on optimizing the shape of the turbine blade and geometric optimization of the draft-tube, of which only future hydroelectric plants will benefit. More efforts should be devoted to improving the design of hydraulic turbines for existing installations.
In a rehabilitation project, the first step is to determine what type of flow downstream of the turbine runner can reduce the energy loss in the existing draft-tube and maximize its recovery efficiency. This analysis is also known as optimizing the inlet speed profile for the draft-tube. The results of solving this optimization problem will be used as a design goal for the new turbine runner.
In a recent study, Galván presented an optimization methodology for draft-tube inlet speed profiles based on an analytical

specification of speed limit conditions at the inlet of a draft-tube cone [4], using a series of commercial softwares. The present paper aims to improve the optimization methodology proposed by Galván, by integrating a more flexible method of representation of the velocity profiles in order to widen the optimization space and by using a more efficient numerical optimization algorithm to speed up the process. Instead of commercial software, this new optimization methodology is implemented with open source counterparts to avoid costly licensing fees. Therefore, the optimization problem can be solved on a larger scale and should be solved faster than before since the maximum number of simulation cases is no longer limited by commercial software licenses.

Method

The flow inside a draft-tube is complex and involves large vortices, recirculation and detachment zones. The solution presented in Fig. 1(left) is for the Porjus U9 draft-tube, which was experimentally and numerically investigated by Mulu et al. [5], and to which the present flow simulations were compared. In order to minimize losses inside such a complex system, the proposed methodology comprises three main components, namely 1) the inlet velocity profiles representation model, 2) the evaluation of the draft-tube performance through CFD simulations, which was validated through comparison with experimental results and 3) the optimization algorithm that modifies inlet velocity to minimize flow losses in the draft-tube. This global process is illustrated in Fig. 1(right), and each component is described below.

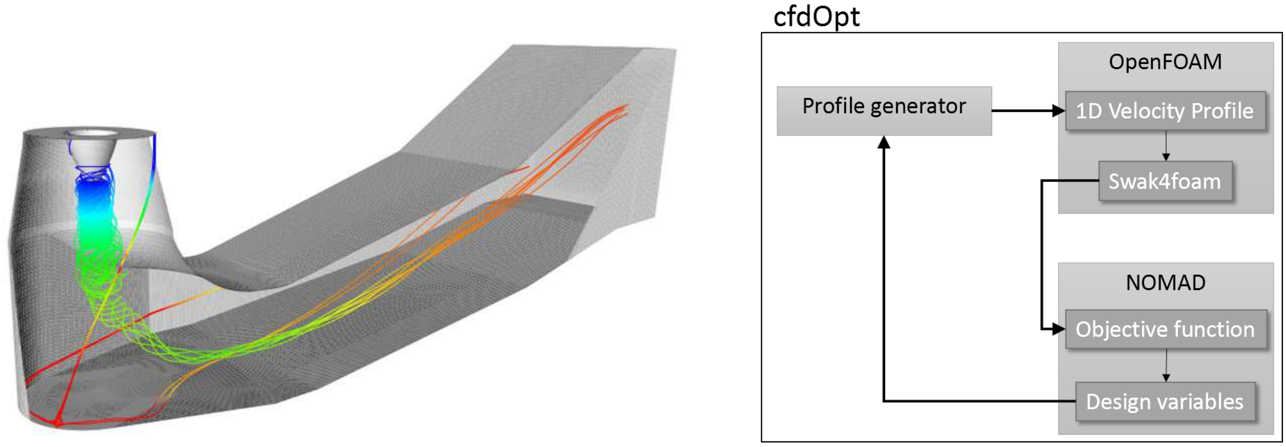

Figure 1: Draft-tube geometry and flow (left) and global optimization process (right)

Velocity Profile Representation

The three-dimensional velocity is represented in a cylindrical coordinate frame, as shown in Fig. 2(left). A typical inlet velocity profile consists of 3 different segments, namely the main profile segment, the inner boundary layer, near the runner hub, and the outer boundary layer, near the draft-tube cone. The main segment is defined as an cubic hermite curve controlled by a sequence of interpolated points. The axial velocity is controlled by five points and the tangential velocity controlled by four points, distributed uniformly along the radius. To reduce the number of free parameters, only y coordinates of the control points are considered as free, the x coordinates being fixed, as illustrated in Fig. 2(right). The radial velocity is given by the following equation:

$$V_r = V_a \cdot \sin(\Theta(r)), \tag{1}$$

where $\Theta(r)$ is the linear combination of the inner hub angle and draft-tube cone opening angle. When the inner hub presence is not considered, the center angle is zero. Both the inner boundary layer segment, near the hub (when present), and the outer boundary layer segment, near the draft-tube wall, are controlled through an analytical power law with a $1/7$ exponent. Boundary layer thickness is fixed explicitly in accordance with flow Reynolds number.

Flow simulations

The Navier-Stokes equations for a Reynolds-averaged incompressible flow is solved with the standard k-ϵ turbulence model, using the simpleFoam flow solver. While the inlet velocity boundary conditions are directly determined through the optimization process, all other boundary conditions are fixed and defined as follows: on walls, no-slip conditions are imposed for velocity, zero-gradient is imposed for pressure and evolutionary wall functions are used for both turbulent kinetic energy and turbulent energy dissipation. At the inlet, a zero-gradient is imposed for pressure, a 5% turbulence intensity is used to compute turbulent kinetic energy and a mixing length of $8,22 \times 10^{-4}$m is used to compute turbulent energy dissipation. At the outlet, a fixed pressure average is imposed, and zero-gradients for all other variables.

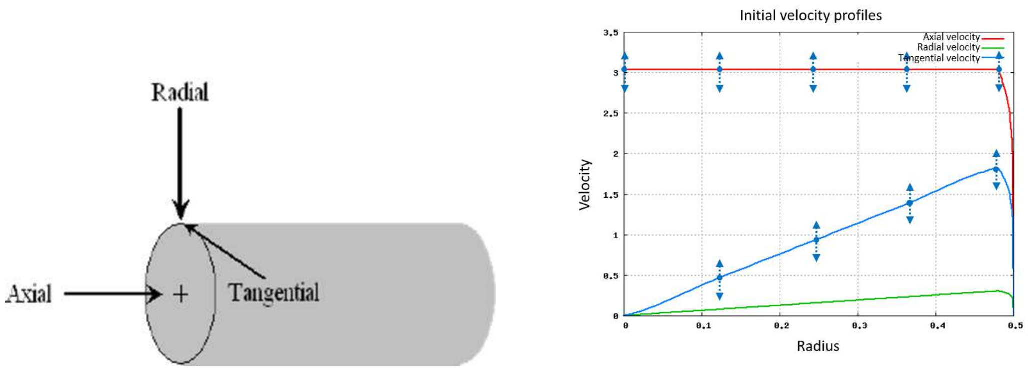

Figure 2: Velocity components in cylindrical coordinates (left) and profile representation (right)

This numerical problem definition allows computing energy losses through the draft-tube, according to the following equation:

$$\zeta = \frac{\frac{1}{A_{in}}\int_{in} P_t dA - \frac{1}{A_{out}}\int_{out} P_t dA}{\frac{1}{2}\rho(\frac{Q}{A_{in}})^2} \quad (2)$$

where P_t represents the total pressure across the inlet A_{in} and outlet A_{out} sections, ρ is the fluid density and Q is the mass flow rate. The energy loss ζ is the objective function that is minimized by the optimization algorithm.
To speed up computations, two criteria are used to determine convergence of the simulations. First, a criterion on pressure and velocity residuals is verified, and second, the energy loss factor must be stabilized.

Optimization algorithm

The optimization algorithm selected in this work is the Mesh Adaptive Direct Search (MADS) algorithm, implemented in the NOMAD open-source software [6]. This algorithm is a generalization of the Generalized Pattern Search (GPS) approach [7], which is a popular gradient-free optimization method that combines a global search step with local polling to refine good candidates and efficiently reach an optimum.
Several parallelization approaches may be used with NOMAD. In the present work, block evaluation mode has been used, where NOMAD executes sequentially, and generates several sets of design variables that can be simultaneously evaluated (see Fig. 3), thereby providing several simultaneous evaluations of the objective function. This execution mode allows taking full advantage of parallelization capacities of the function evaluations, which, in the present case, are performed through OpenFOAM simulations.

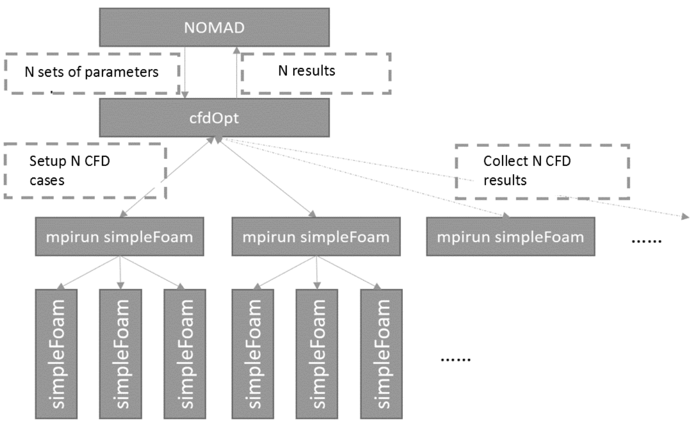

Figure 3: Optimization problem parallelization approach using NOMAD

Optimization results

Several verification and validation cases have been performed to assess the quality of flow solutions and convergence of the optimization algorithm. We only present here a sample of the optimization results obtained for the Porjus U9 test case. A preliminary optimization step consisted in determining an optimal inlet boundary condition based on a solid body rotation profile. This initial profile, illustrated in Fig. 2(right) yielded an energy recovery factor of 0,1585, which constitutes an initial loss reference to compare the optimization results with. Figure 4 illustrates the computational mesh used (top left) and the pressure distribution across the domain (bottom left) for the optimal inlet boundary condition reached, which is illustrated in Fig. 4(right). The energy recovery factor for this optimal profile is 0,0565, which is a 64.3% reduction of draft-tube losses.

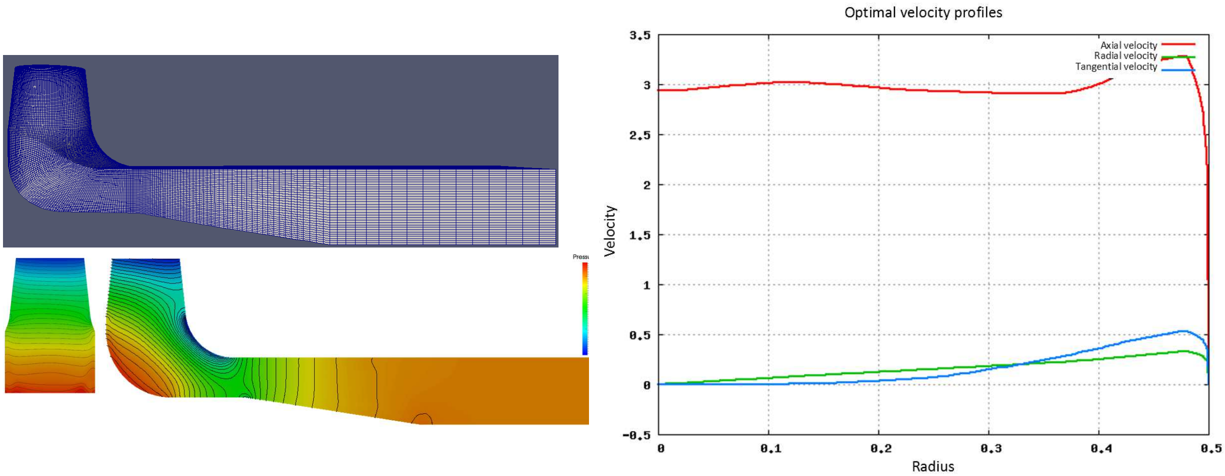

Figure 4: Draft-tube mesh (top-left), optimal pressure field (bottom-left) and optimal velocity profile (right)

Conclusion

This paper has presented a methodology to formulate and solve an inlet velocity profile optimization problem to minimize hydraulic turbine draft-tube losses. The proposed approach, based on the Mesh Adaptive Direct Search (MADS) optimization algorithm coupled to an incompressible RANS CFD simulation, uses OpenFOAM and the standard $k - \epsilon$ turbulence model. The methodology was tested on several test cases, and results are presented for one condition of the Porjus U9 draft-tube. The results show that the energy loss factor was reduced by more than 60% in optimization cases compared with the best efficiency point found using a solid body rotation test. These optimization results can be used as a design reference for turbine designers working on rehabilitation projects of hydraulic power plants.

References

[1] U. Andersson and N. Dahlbäck, “Experimental evaluation of draft tube flow. A test case for CFD simulations,” in *XIX IAHR Symposium on Hydraulic Machinery and Systems*, 1998.

[2] S. Galván, M. Reggio, and F. Guibault, “Optimization of the inlet velocity profile in a conical diffuser,” in *ASME 2012 Fluids Engineering Division Summer Meeting*. American Society of Mechanical Engineers, 2012, pp. 125–134.

[3] S. Bahrami, C. Tribes, S. von Fellenberg, T. C. Vu, and F. Guibault, “Physics-based surrogate optimization of Francis turbine runner blades, using mesh adaptive direct search and evolutionary algorithms,” *International Journal of Fluid Machinery and Systems*, vol. 8, no. 3, pp. 209–219, 2015.

[4] S. Galván, M. Reggio, and F. Guibault, “Numerical optimization of the inlet velocity profile ingested by the conical draft tube of a hydraulic turbine,” *Journal of Fluids Engineering*, vol. 137, no. 7, pp. 071 102–071 102–15, 2015, 10.1115/1.4029837.

[5] B. Mulu and M. Cervantes, “Experimental investigation of a Kaplan model with LDA,” in *Proceedings of the Water Engineering for a Sustainable Environment Congress: 33rd IAHR Congress, Vancouver, Canada*, 2009, pp. 155–162.

[6] M. A. Abramson, C. Audet, J. E. Dennis Jr, and S. Le Digabel, “OrthoMADS: A deterministic MADS instance with orthogonal directions,” *SIAM Journal on Optimization*, vol. 20, no. 2, pp. 948–966, 2009.

[7] C. Audet and J. E. Dennis Jr, “Analysis of generalized pattern searches,” *SIAM Journal on Optimization*, vol. 13, no. 3, pp. 889–903, 2002.