Optimization procedure to design a Francis turbine runner using 2D Through-flow analysis

Ryo Watanabe¹, Kazuyoshi Miyagawa², Mohammad Hossein Khozaei², Naoki Yamaguchi³, Toshitake Masuko¹

¹Meidensha Ltd, 2-8-1, Osaki, Shinagawa-ku, Tokyo, 141-8565, Japan
²Department of Applied Mechanics, Graduate School of Mechanical Engineering, Waseda University
³EAML Engineering Co., LTD, Japan

Abstract. Due to increasing usage of Feed-in Tariff (FIT), which is a subsidy system to encourage expansion of renewable energy and to reduce costs associated with introduction of renewable energy, demand for hydraulic turbine designers has increased solely for the purpose of improving turbine performance. With these requirements, CFD simulation is a powerful tool for designers to improve the internal flow and pressure distribution in turbines. Traditionally, most designers use a trial-and-error process based on previous experience to design new turbines. Usually, achieving the goal is a lengthy and painstaking process. This paper proposes a novel method which combines both CFD and optimization methods to automatically optimize a Francis turbine. This procedure includes four components: a design program, a CFD solver, a scheduler, and an optimization algorithm. This paper describes an attempt to automatically optimize a Francis turbine runner by using ANSYS Vista TF as the 2D CFD solver based on through-flow theory, and ANSYS CFX 19.1 as the 3D CFD solver based on Finite Volume Method (FVM). The runner geometry is parameterized by Bezier curves in the design program. CFD analysis is performed to assess the efficiency of the runner, and also the output power is calculated by runner torque. These two parameters are used as the objective function of the optimization algorithm. This method is used to optimize the runner geometry, which consists of two parts: the meridional plane and profile camber lines. We confirmed that the new turbine, which was optimized by the above-mentioned method, has better efficiency than the existing one.

Nomenclature

| Symbol | Definition |
|--------|------------|
| d      | diameter [m] |
| F_d    | dissipation force [N/kg] |
| h      | static enthalpy [J] |
| H      | specific enthalpy [J/kg] |
| m      | distance along meridional direction [m] |
| M      | non – dimensional distance along meridional direction [-] |
| p      | pressure [Pa] |
| r      | radius [m] |
1. Introduction
Continuous efforts are being made to improve the performance of hydraulic turbines. As computer performance increases, new shapes are made using CFD. In the case of Japan, requests for turbine designs are increasing, but designing turbines takes excessive time and turbine performance depends on designer skill and experience. In recent years, therefore, many papers about turbine optimization methods using 3D CFD analysis, have been published. 3D CFD analysis has been performed to obtain more detail about the inner flow, pressure, and velocity distribution in static and dynamic domains, but it takes a long time to obtain results in one case analysis. From the designer’s point of view, it is difficult to use 3D CFD as the optimization system component because of time constraints and the complexity of systems which combine many tools. This paper presents an optimization method that uses a combination of 2D and 3D CFD analysis. Optimization is ordered has follows: First, obtain the optimization shape through 2D CFD analysis. Second, confirm the performance of the optimized shape using 3D CFD analysis. Third, compare the performance of conventional shape with that of the optimized.

2D CFD analysis is based on the through-flow code, which is used to rapidly arrive at the initial design of turbo machinery, especially for compressors [1]. The method, however, has not been applied to hydraulic turbines. The present study therefore focuses on the results of a design system that includes the through-flow code. This paper compares the results of conventional database design and the new design, based on an optimization system using a 2D and 3D CFD solver.

2. Streamline curvature Through-flow
Many publications derive the equations for the streamline curvature through-flow method, so only an overview is given here.

The key to the through-flow calculation method is the axisymmetric solution of the radial equilibrium equation in the $S_2$, meridional surface. This equation will be derived starting from those for steady, inviscid flow relative to the rotor (figure 1).

The equations solved are the continuity equation, equation (1), and the energy equation, equation (2), which explain a combination of the first law of thermodynamics and the Euler equation of turbo machinery [4], a suitable equation of state, and the inviscid momentum equation, equation (3), (4), (5), for the flow on the mean stream surface $S_2$ in the form of the general radial equilibrium equation, equation (6).
To calculate by computer, the equation must be discretized. For through-flow code, create a grid and calculation of the equation at the point, which separates the streamline by grid. The grid for the calculation is created by streamline, which is determined from the number of separated meridional flow paths and the number of fixed calculating stations, which are almost vertical to channel walls.

\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x} (\rho v_x) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v_r) + \frac{1}{r \partial \theta} (\rho v_{\theta}) = 0 \]  
\[ I = h + \frac{1}{2} (w^2 + u^2 + 2uw_y) - u \cdot (w_y + u) = h + \frac{1}{2} w^2 - \frac{1}{2} u^2 \]
\[ \text{Axial Momentum:} \quad v_r \nabla v_r = -\frac{1}{\rho} \frac{\partial \rho}{\partial z} \]
\[ \text{Tangential Momentum:} \quad v_r \nabla v_r + \frac{v_\theta}{r} v_r + 2\omega v_r = -\frac{1}{\rho r} \frac{\partial \rho}{\partial \theta} \]
\[ \text{Radial Momentum:} \quad v_r \nabla v_r - \frac{v_\theta^2}{r} v_r - r \omega^2 + 2\omega v_\theta = -\frac{1}{\rho} \frac{\partial \rho}{\partial r} \]

where \( v_r = v_r, \frac{\partial}{\partial z} + v_r \frac{\partial}{\partial r} + \frac{v_\theta}{r} \frac{\partial}{\partial \theta} \)

\[ \frac{dv_r}{dq} = \frac{dH}{dq} - T \frac{ds}{dq} - \frac{1}{2r^2} \frac{d(r^2 v_\theta^2)}{dq} + \sin \phi \frac{v_\theta^2}{r} + \cos \phi v_\theta \frac{v_\theta}{cm} + \tan \theta \frac{v_\theta}{r} \frac{\partial (r v_\theta)}{\partial m} + F_d \]  

3. Automated design system
The automated design system described in this paper contains bellow elements and flow (figure 2). The automatic design method using 3D CFD is already existed [5], [6], but the figure shows the system, which combined the 2D CFD and 3D CFD, and consists of below characteristics.
- A pre design (meridional profile and curve profile) is based on Database.
- A new parameter is based on the geometry definition system which changes the Bezier profile.
- Geometry data are transferred from the software, which can change the blade profile (ANSYS BladeModeler) to the solver (ANSYS Vista TF) that is based on through-flow code.
- An optimized design is evaluated by the 2D and 3D CFD solver (ANSYS CFX)
- The optimization method is given by FAST and pileOPT

Considerations with regard to the definition of the most suitable objective function based on experience of designer.

**Figure 2.** The procedure of optimization. This optimization starts from the database design as the first design element. Through optimization, flow and change the blade shape. The results of the designed shape satisfy design conditions, when the optimization flow is concluded.

**4. Geometry definition methods**

The runner shape is defined in 2 profiles. First, the meridional plane, which is made by projecting the blade surface in the direction of rotation, as given by the database, below figure shows the image of meridional plane made by database and horizontal axis shows the distance of direction of rotating axis, and vertical axis shows the distance of direction of radius (figure 3). Second, the curve profile, that includes the beta and theta distributions. The curve profile is the main target of optimization in this paper.

The curve profile from each 5 span (Crown to Band) is defined by several control points. The points control the distribution of beta and theta, and distribution is made by the method of interpolation using some discrete points. This beta distribution makes the camber line of the blade shape at each span.

**Figure 3.** Meridional Plane of middle specific speed given by database
5. Optimization method

The blade shape is defined by camber line, with the line interpolated by distribution of beta. This paper shows how to make the camber line in optimized flow.

First, the 4 points were set, as in the figure below, the horizontal axis shows the meridional distance, and vertical axis shows the blade angle (figure 4). Only 2 points (placed in middle) can move in the range; this range is determined by the designer, based on experience. Second, the camber line is drawn as a Bezier curve to interpolate the 4 points as calculated using the equation (7). Third, the line is separated one-tenth at a discrete point and the coordinate data of the points is given to the modeler.

![Bezier Curve and Control Points](image)

\[ \beta = (1 - M)^3 \cdot \beta_0 + 3 \cdot (1 - M)^2 \cdot M \cdot \beta_1 + 3 \cdot (1 - M) \cdot M^2 \cdot \beta_2 + M^3 \cdot \beta_3 \] (7)

6. Optimization function

Ideally it would be best to maximize the efficiency or minimize the losses in the runner region. This procedure, however, would require too much time and inappropriately constrain conditions and objective functions. In cases using many input parameters and output parameters and when the design system is given a correct direction of optimization, we can obtain ideal results. This method, however, would take too much time and the systems often tend to be complicated.

In this paper, we set, the following parameters to optimize the blade shape.

**Input parameters**
- The points, these include β and M coordinates, to control beta at each span
- The points, these include β and M coordinates, to control Bezier curves at each span

**Constraint Conditions**
- The points, these include the \( \theta \) coordinate, of TE at each span
- Throat width and position at each span

**Output and objective parameters**
- Turbine output
- Efficiency

The optimized objective gives the direction to the Turbine output and Efficiency increase and throat width beyond design conditions and throat positions beyond \( M = 0.85 \). Also the condition is set that, if
the throat width is below design conditions and the throat position is located below \( m=0.85 \), the results would eject an error value and the results would be ignored.

The present study using two optimization algorithms, FAST and pileOPT, which are developed by ESTECO Ltd. The profiles of these two algorithms are as follows:

**FAST**
- This method combines the virtual and real optimization algorithms.
- This method is applicable to both single or multi-objective function problems.
- This method is in an early stage, FAST is used for training the DOE strategy and creating the database.

**pileOPT**
- This is a multi-strategy and self-adaption algorithm; the method combines local and global searches.
- This method is applicable to single or multi-objective function problems.
- This method provides automatic assessment of the ratio of virtual and real design performance.
- This method is powerful in heavy simulations.

First, start the optimization using FAST. Second, use pileOPT based on FAST results. Third, assess both results to see if they obtain similar distribution and conversion.

### 7. RESULTS OF OPTIMIZATION

**Table 1.** Comparison of original and optimized 2D analysis results

| Design               | Original | Optimized |
|----------------------|----------|-----------|
| Turbine output [kW]  | 10.76    | 9.4       |
| Head t-t [m]         | 3.96     | 3.44      |
| Head Coefficient     | 1.46     | 1.27      |
| Isentropic Efficiency t-t [%] | 77.30 | 78.04 |

**Table 2.** Comparison of original and optimized 3D analysis results

| Design               | Original | Optimized |
|----------------------|----------|-----------|
| Relative Runner Head [-] | 1.00  | 0.95    |
| Relative Turbine output [-] | 1.00  | 0.96    |
| Relative Runner Efficiency [-] | 1.00  | 1.01    |
Figure 5. Comparison of rothalpy loss distribution at runner pressure and suction surface. The reference region is the runner inlet. The upper surfaces are original, and surfaces below are optimize.

\[ \text{Loss} = \frac{I_{\text{ref}} - I_t}{\frac{1}{2} u_{\text{ref}}^2} \] (8)

The optimization results give the difference of efficiency and power value (table 1). The original has more turbine output, but worse efficiency. The head consumed at the runner region in the original is higher than in the optimized. The efficiency and power show a good relation in both 2D and 3D results. In this case, the efficiency of the optimized method is improved by 1.0 over than original and the loss distribution is also better than in the original.

The contour shows the distribution of loss, which calculated based on equation (8), at runner pressure side and suction side as a meridional plane (figure 5). The optimization method decrease the loss at mid flow path to TE flow path on the surface of the pressure side and hub to mid span on the surface of the suction side. The reduction of the loss at each side make improvement of the efficiency as a results.

8. Conclusion
The results show that efficiency is improved, but turbine output is worse. The through-flow code analysis gives us rapid results, but further, more detailed analysis, is required to check the reliability of the method. Key aspects of this method are its rapidity, its easy settings, the possibility of obtaining results for nearly all shapes and performance levels of design, and finally ability to optimize by hand using complex design tools. As a future work we will construct the system combine the through-flow code and 3D CFD code.

References
[1] Michael Casey, Frank Gersbach, Chris Robinson 2008 AN OPTIMIZATION TECHNIQUE FOR RADIAL COMPRESSOR IMPELLERS (ASME)
[2] Chung-Hua Wu 1952 A general theory of three-dimensional flow in subsonic, and supersonic turbomachines of axial, radial and mixed-flow types (Washington, NACA)
[3] Michael Casey, Frank Gersbach, Chris Robinson 2008 A NEW STREAMLINE CURVATURE THROUGHFLOW METHOD FOR RADIAL TURBOMACHINERY (ASME)
[4] S.L. Dixon, B.Eng. 1998 Fluid Mechanics Thermodynamics of Turbomachinery (Elsevier) p.23-54
[5] Jingchun Wu, Katsumasa Shimmei, Kiyohito Tani, Kazuo Niikura, Joushirou Sato 2007 CFD-
Based Design Optimization for Hydro Turbine (ASME Journal of Fluids Engineering)

[6] Kazuyuki Nakamura, Sadao Kurosawa 2009 Design Optimization of a High Specific Speed Francis Turbine Using Multi-Objective Genetic Algorithm (International Journal of Fluid Machinery and Systems Vol. 2, No. 2)

Notification
The ANSYS notation is a registered trademark of ANSYS Inc.