

# A simple inverse design method for pump turbine

Junlian Yin<sup>1</sup> Jingjing Li<sup>1</sup> Dezhong Wang<sup>1</sup> Xianzhu Wei<sup>2</sup>

<sup>1</sup>School of mechanical engineering, Shanghai Jiao Tong University, 200240, China

<sup>2</sup>State Key Laboratory of Hydropower Equipment, Harbin 150040, China

Email: jlyin@sjtu.edu.cn

**Abstract.** In this paper, a simple inverse design method is proposed for pump turbine. The main point of this method is that the blade loading distribution is first extracted from an existing model and then applied in the new design. As an example, the blade loading distribution of the runner designed with head 200m, was analyzed. And then, the combination of the extracted blade loading and a meridional passage suitable for 500m head is applied to design a new runner project. After CFD and model test, it is shown that the new runner performs very well in terms of efficiency and cavitation. Therefore, as an alternative, the inverse design method can be extended to other design applications.

## 1. Introduction

Basically, there are two approaches applied in the hydraulic design of pumps or turbines[1, 2]. One is based on geometrical parameters and the other is based on hydrodynamic parameters, i.e. blade loading distribution (BLD) or prescribed pressure distribution. The latter approach is defined as inverse design, in which the blade shape and flow field can be obtained simultaneously. One of the advantages for inverse design is that the design guidelines or design expertise thus obtained are expected to be more universal and operator-independent, and are easily transferred to the next generation[3]. In other words, the design inputs, which are obtained by one precious hydraulic model with excellent performances, can be extended into other design works. In this sense, the inverse design is featured with inheritance. According to state of the art of inverse design, three-dimensional design method has been well established by Zangh[4, 5] and Goto[6, 7] et al. and widely applied in design practices. Due to the mathematical rigor of 3D inverse design theory, it should be viable if we put the BLDs extracted from a perfect hydraulic model into other designs with similar specific speeds. To do this, CFD work is necessary for extracting of BLDs. To avoid the extra CFD work, a simpler method is proposed in this study to replace CFD to obtain BLD. And also, a corresponding simple inverse design which can incorporate the above BLD as input is developed. The systematic approach was implemented into a pump turbine runner design.

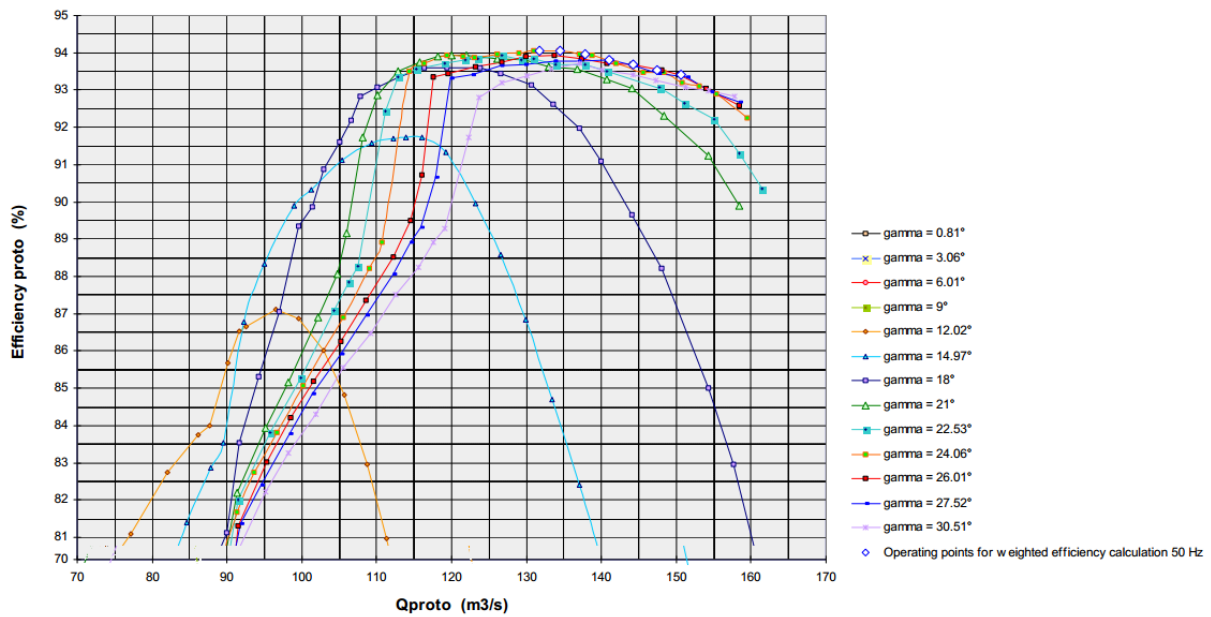
## 2. Design target

The reference runner is designed for a high specific speed pump turbine, which is served for a 200m pump storage plant. The expected prototype performance calculated from the model test data is illustrated in Fig.1, from which the calculated weighted average efficiency in pump mode is 93.84%. The parameters involved in the reference design are listed in Table.1 with specifications of the new design. The objective design is designated for 500m pump storage plant. How to utilize the spirit of the reference runner to design the new one is the focus of this study.



**Table.1** Comparison of design inputs between two models with different specific speeds

	Reference runner	New runner
Head	206	473
Flow rate	140	72.25
Rotation speed	250	375
Specific speed	198.6	114.75
Prototype efficiency	93.86	?

**Fig.1** Efficiency profiles for various guide vane openings for the prototype reference pump turbine

### 3. Design methodology

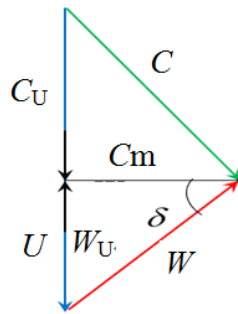
#### 3.1. Direct analysis of the reference runner

The intention for direct analysis is to obtain the BLD, one representation of which is the Euler energy. From the basic Euler equation (Eq.1) describing the relations between the head and velocities, the Euler energy can be defined as the product of peripheral velocity and tangential component of absolute velocity.

$$Hg = U_2 C_{u2} - U_1 C_{u1} \quad (1)$$

According to the velocity triangle as shown in Fig.2, Eq.1 can be written as:

$$UC_u = U(U - C_m \tan \delta) \quad (2)$$



**Fig.2** Velocity triangle used in the direct analysis

The next step is to calculate the meridional velocity  $C_m$  and the blade angle  $\delta$ . In theory, the flow field is three dimensional and three velocity components are coupled with each other. Only the 3D CFD approach can deal with the exact solution. To avoid the troublesome cost due to CFD application, the following simplicity assumptions are made.

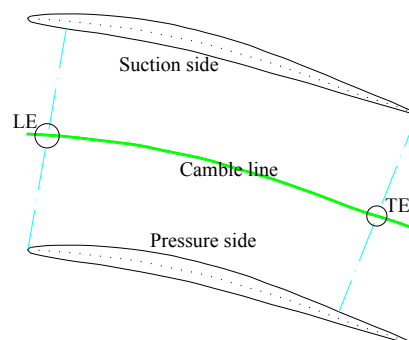
The volume averaged Euler energy for the flow passage can be represented by the camber of the blade, as shown in Fig.3. In this way, the blade angle  $\delta$  can be computed by analysis on geometrical parameters of the reference blade. Fig.4 illustrates the relations between the blade angle and relevant parameters. The Cartesian coordinates of the 3D blade  $(X, Y, Z)$  were transferred into combined representation with meridional length and arc length in the peripheral direction  $(M, R)$  by Eq.3, Eq.4 and Eq.5. By doing so, the blade angle was calculated by Eq.6.

$$\Delta L = \sqrt{\Delta X^2 + \Delta Y^2} \quad (3)$$

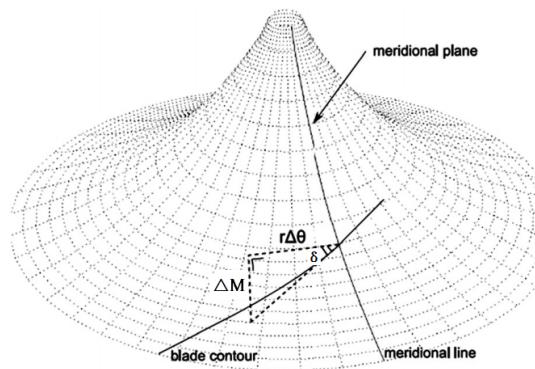
$$\Delta M = \sqrt{\Delta L^2 + \Delta Z^2} \quad (4)$$

$$\Delta \theta = a \tan \left( \frac{X_i}{Y_i} \right) - a \tan \left( \frac{X_{i-1}}{Y_{i-1}} \right) \quad (5)$$

$$\delta = \text{atan} (r \Delta \theta / \Delta M) \quad (6)$$

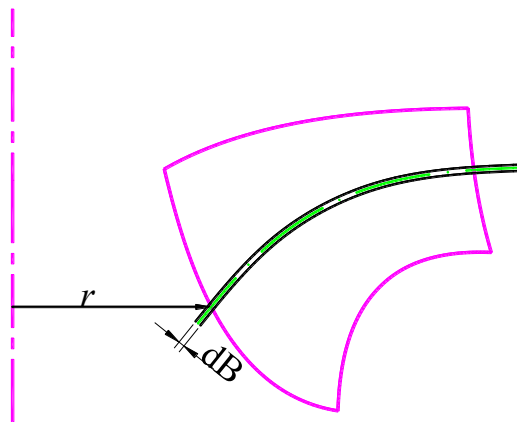


**Fig.3** The camber line used to represent the averaged Euler energy of the flow passage



**Fig.4** The geometrical relations between blade angle and relevant parameters

It was assumed that the flow in the meridional plane was essentially two-dimensional, and that the effects of the velocities (and the gradients in the velocity or pressure) normal to the meridional surface were negligible. Moreover, it was tacitly assumed that the flow in a real turbomachine could be synthesized using a series of these two-dimensional solutions for each meridional annulus[8]. In doing so it is implicitly assumed that each annulus corresponds to a streamtube such as depicted in Fig.5. If the radius  $r$  and the thickness  $dB$  are known a priori, the meridional velocity for each streamtube can be obtained. For simplicity, the thickness was computed based on equal division in the spanwise direction. In addition, the squeezing effect due to blade thickness can easily be taken into account.

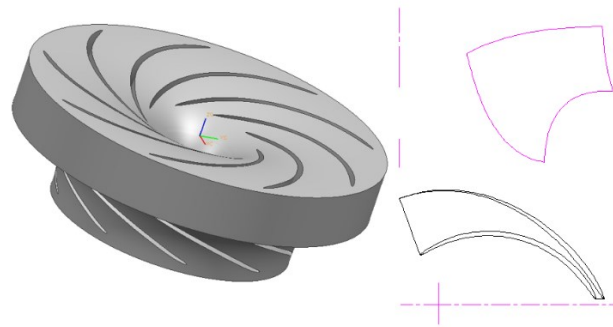


**Fig.5** The streamtube in the meridional plane for meridional velocity calculation

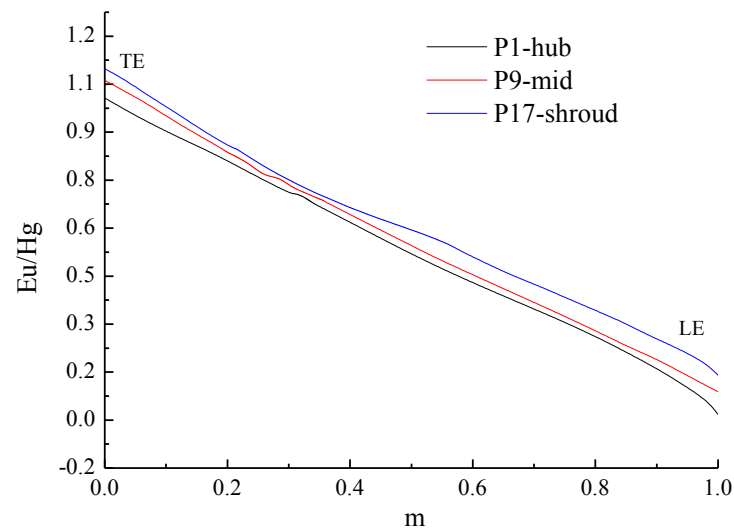
The above two assumptions give:

$$Eu = UC_U = \frac{2\pi nr_i}{60} \cdot \left( \frac{2\pi nr_i}{60} - C_m \psi \tan \delta \right) \quad (7)$$

Up to here, the direct problem was established in a very simple way with some reasonable assumptions. The well posed analysis method was first implemented into the reference runner (shown in Fig.6), which is served for a 200m pump turbine. The results are shown in Fig.7, which implies that the Euler energy distribution follows some kind of principle. The regularity indicates the assumptions made in the direct analysis is of rationality.



**Fig.6** Reference runner with the meridional passage and projected drawing



**Fig.7** The Euler energy distribution of the reference runner based on the direct analysis method

### 3.2. Inverse design

For purpose of inverse design, a reverse process, compared with the direct analysis, is established. Given the flow rate, rotation speed and head, we choose a reference meridional passage with similar specific speed, the BLD profiles and blade thickness distribution, the 3D blade geometry will be figured out. In this way, only the BLD is the key variable for optimization design. Since the focus of this study is checking the heritage effect of inverse design, the BLD obtained above is directly taken as an input for the new runner.

For the new design, a reference meridional passage is selected and shown in Fig.8. Substituting the BLD in Fig.7 and the design specifications listed in Table.1 into the existing meridional passage brings out the 3D blade shape.

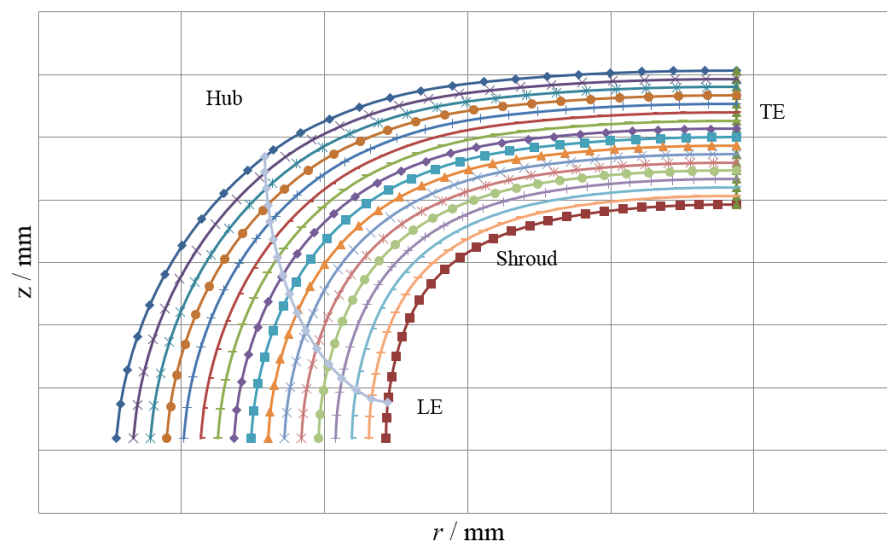


Fig.8 The meridional passage selected for the 500m pump turbine

#### 4. CFD Results and Model Test

To validate the hydrodynamic design, the CFD method was used to predict the performance of the pump turbine in pump mode[9, 10]. The flow field was calculated by solving the three-dimensional steady incompressible Reynolds Averaged Navier Stokes (RANS) equations with the RNG k- $\epsilon$  turbulence model[11] using the commercial software Ansys CFX 12.1.

As shown in Fig.9, the computational domain embracing the whole flow passage, which consists of the spiral casing, the stay vanes, guide vanes, runner blades, and the draft tube. The static pressure and rated mass flow rate were set as the boundary conditions for the inlet and outlet of the computational domain, respectively. Water was considered to be the working fluid, and the solid surfaces were considered to be hydraulically smooth with no-slip. For the connection between the rotating runner and the adjacent vanes, as well as the runner and draft tube, the stage method was used.

A hybrid grid system was constructed in the computational domain, with hexahedral elements filling the runner domain and the draft tube domain and tetrahedral elements in the other domains. 8 layers of prism elements were generated to refine the grid density near the walls of the guide and stay vanes. A grid-dependency test was carried out with various numbers of grids, and the optimum number of grids was selected as approximately 4,000,000 grid nodes.

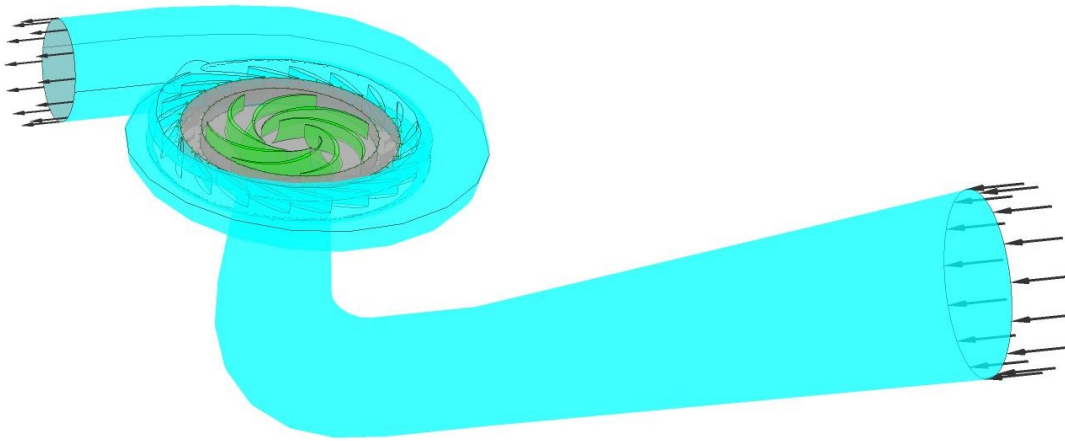
In the computation, RMS residual values of the momentum and mass were set to fall below  $1.0\text{E-}04$ . The physical time scale was set to  $0.1/\omega$ , where  $\omega$  is the angular velocity of the runner. The converged solutions were obtained after approximately 400 iterations.

The CFD study of the whole flow passage under the rated flow condition with the incident flow angle at the inlet of the guide vane is zero was carried out. The results shown that the hydraulic efficiency is 92 for model pump turbine, in which the size of inlet radius is 270mm.

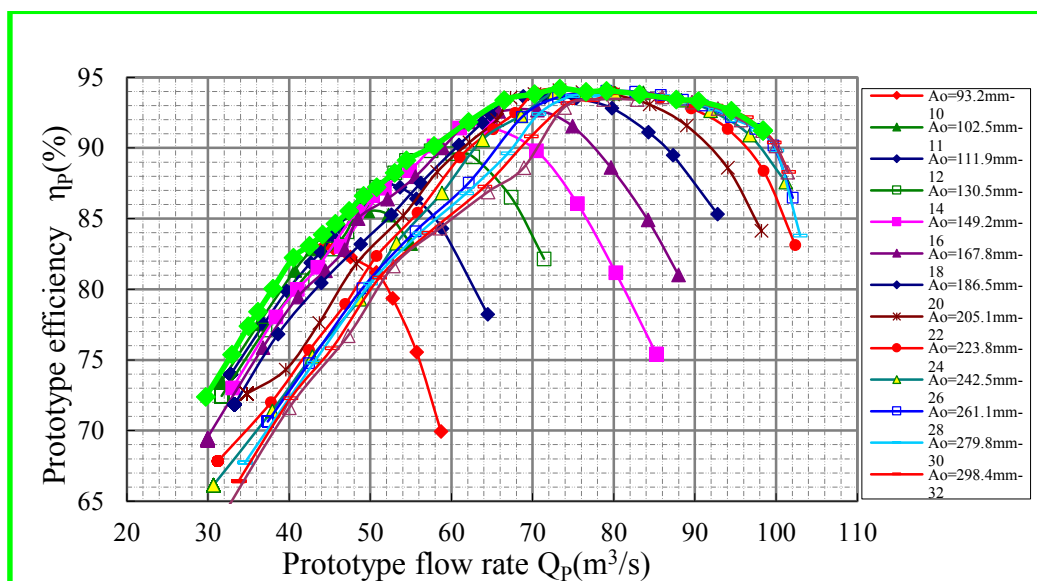
After the CFD validation for the runner, Tests were carried out on the model to, again, validate its performance in the high-head test rig at the Harbin Institute of Large Electric Machinery in China[12]. The experimental data for the pump mode under various guide vane openings are illustrated in Fig.10. Based on the weighted averaged calculation and conversion regulations based on IEC60193-1999, the averaged efficiency is 94%. When compared with the reference design, which efficiency is 93.86, the performance of the new design can be comparable.

Thus far, the combination of proposed direct analysis and the inverse design approach can be proven to be an effective method for hydraulic design engineering. From the point view of flow physics, error is certainly present in the direct analysis method. However, when integrated with the corresponding

inverse design method, the error is digested naturally. In this sense, the heritage of the spirit from one excellent hydraulic model to another is realized.



**Fig.9** CFD modeling of the new design



**Fig.10** The converted prototype test data in pump mode for the new design

## 5. Conclusion

In this paper, a systematic approach, which combines a direct analysis method and a simple inverse design method, was proposed for hydraulic design practice, in which the spirit of one reference hydraulic model with excellent performances was transferred into a new hydraulic model with different specific speeds.

Based on two assumptions in the direct analysis process, the Euler energy distribution of the camber line, which represents the blade passage, was extracted. The profile exhibits a very regular energy transformation from the leading edge to the trailing edge. When implementing the known BLD into a new meridional passage, a new blade with geometrical smoothness was obtained. After CFD and model test validation, the performance of the new design, specifically the efficiency, is superior.

In this sense, the proposed design approach can be taken as an effective way for hydraulic design engineering.

### Acknowledgement

The research work was funded by China Postdoctoral Science Foundation (2013M531173), Shanghai Postdoctoral Science Foundation (13R21414100). Authors are grateful to the staff from Harbin Large Electric Machinery, China, for his contribution to this work

### References

- [1] Wu J C, Shimmei K, Tani K, et al. 2007, CFD-based design optimization for hydro turbines[J]. *Journal of fluids engineering-transactions of the ASME*, **129**(2): 159-168.
- [2] Borges J E. 1990, A three-dimensional inverse method for turbomachinery: Part 1-theory[J]. *Journal of Turbomachinery*, **112**(3): 346-354.
- [3] Zangeneh M. 2007, Advanced design software for pumps[J]. *World Pumps*, **489**: 28-31.
- [4] Zangeneh M, Goto A, Harada H. 1998, On the design criteria for suppression of secondary flows in centrifugal and mixed flow impellers[J]. *Journal of turbomachinery-transactions of the ASME*, **120**(4): 723-735.
- [5] Zangeneh M, Goto A, Harada H. 1999, On the role of three-dimensional inverse design methods in turbomachinery shape optimization[J]. *Proceedings of the institution of mechanical engineers part C-journal of mechanical engineering science*, **213**(1): 27-42.
- [6] Goto A and Zangeneh M. 2002, Hydrodynamic design of pump diffuser using inverse design method and CFD[J]. *Journal of fluids engineering-transactions of the ASME*, **124**(2): 319-329.
- [7] Goto A, Nohmi M, Sakurai T, et al. 2002, Hydrodynamic design system for pumps based on 3-D CAD, CFD, and inverse design method[J]. *Journal of fluids engineering-transactions of the ASME*, **124**(2): 329-335.
- [8] Christopher E B. 1994, *Hydrodynamics of Pumps*[M]. Oxford: Oxford University Press.
- [9] Yin J L, Liu J T, Wang L Q, et al. 2010, Performance prediction and flow analysis in the vaned distributor of a pump turbine under low flow rate in pump mode[J]. *science china-technological sciences*, **53**(12): 3302-3309.
- [10] Wang L Q, Yin J L, Jiao L, et al. 2011, Numerical investigation on the "S" characteristics of a reduced pump turbine model[J]. *Science china-technological sciences*, **54**(5): 1259-1266.
- [11] Orszag S A, Yakhov V, Flannery W S, et al., *Renormalization Group Modeling and Turbulence Simulations*[Z]. Tempe, Arizona.
- [12] Yin J L, Wang D Z, Wei X Z, et al. 2013, Hydraulic improvement to eliminate S-shaped curve in pump turbine[J]. *Journal of fluid engineering-ASME*, **135**(7): 71101-71105.