Design optimization of hydraulic turbine draft tube based on CFD and DOE method

Mun Chol Nam[1], **Ba Dechun**[1,*], **Yue Xiangji**[1], **Jin Mingri**[1]

[1]School of Mechanical Engineering & Automation, Northeastern University, Shenyang 110819, China;

Corresponding author: Ba Dechun, E-mail: dchba@mail.neu.edu.cn

Abstract. In order to improve performance of the hydraulic turbine draft tube in its design process, the optimization for draft tube is performed based on multi-disciplinary collaborative design optimization platform by combining the computation fluid dynamic (CFD) and the design of experiment (DOE) in this paper. The geometrical design variables are considered as the median section in the draft tube and the cross section in its exit diffuser and objective function is to maximize the pressure recovery factor (C_p). Sample matrixes required for the shape optimization of the draft tube are generated by optimal Latin hypercube (OLH) method of the DOE technique and their performances are evaluated through computational fluid dynamic (CFD) numerical simulation. Subsequently the main effect analysis and the sensitivity analysis of the geometrical parameters of the draft tube are accomplished. Then, the design optimization of the geometrical design variables is determined using the response surface method. The optimization result of the draft tube shows a marked performance improvement over the original.

1. Introduction

The efficiency of the hydraulic turbine is greatly influenced by the performance of the draft tube, the main role of the draft tube is to convert the remaining kinetic energy into the static pressure at the runner outlet which is greatly affected by the performance of the draft tubes in hydraulic turbine, the performance optimization can be achieved by varying the size and shape of draft tube. The draft tube can be classified into two groups; the straight and curved shapes. Generally, the straight shape of the draft tube has a good hydraulic characteristic, but it is used only at the small and medium diameters of the runner, because of the huge construction cost of the long vertical draft tubes. While the curbed shape of the draft tube is used at the hydraulic turbine with the large diameter of the runner to reduce the excavation depth of the draft tube. Traditionally, the design of the draft tubes has been performed through the experience of the designers and the model tests.

Many investigations have been carried out by using computation fluid dynamic (CFD) for the performance calculation, the analysis and the optimization design of the draft tubes in the hydro power plants. V Prasad et al. [1] studied on the optimal design by varying the geometric parameters such as the length and height of the elbow of the draft tube at different mass flow rate using 3D viscous flow simulations, in which the geometrical parameters of the best performance from numerical simulation were close to height ratio of 2.24 and length ratio L/D1 of 6.0. V Soni et al. [2] investigated the effects of the suction cone height in the draft tube on the performance of Francis turbine. The results of the numerical simulations using CFD were reported, in which the several permutation and combination of the geometrical parameters such as the suction cone, elbow and exit diffuser were achieved to improve
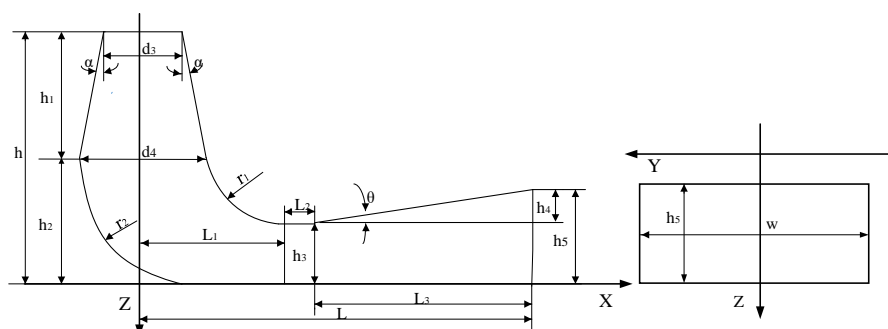
the performance of the curved draft tube. R Khare et al. [3] performed on the optimization design by varying the geometric parameters of the length and diffuser angle of the conical draft tube, the result indicates that most of the hydraulic turbines with the straight conical draft tube are achieved close to result which the diffuser angle is 3.6º to 6º and the length is 19D_3. B D Marjavara et al. [4] carried out the optimization design of the Turbine-99 draft tube using response surface methodology (RSM), in which the curvature radius of the draft tube was taken as variable and the average pressure recovery factor and the energy loss factor were taken as the objective function. The computed result showed that the optimization results were similar to the experimental, confirming the application possibilities of these techniques in optimal design process. Meanwhile, because the process of the shape optimization of the draft tube is realized by many iterative calculations using CFD, it requires many time and manpower. Therefore, the automatic optimization methods using the geometrical parameters of the cross-sections of the draft tube have been examined [5, 6, 7], but these methods have not considered the geometrical parameters such as the median section affecting significantly on the performance of the draft tube, the geometrical parameters of its meridian channel are the most significant ones affecting on the performance. This paper presents the automatic solving model which integrated the geometric module and numerical calculation module based on the multi-disciplinary platform. The median section and the cross section of the exit diffuser of the draft tube are selected as the geometrical design variables, and the objective functions are selected as maximizing of the pressure recovery factor (C_p). The main effect analysis, the sensitivity analysis and the interaction analysis of the draft tube in hydraulic turbine are accomplished by the calculation of CFD and the design of experiment (DOE) based ISIGHT platform, and the design optimization is determined using the response surface method.

2. Design parameters and objective function

2.1 Design parameters

One of the most important problems in the optimal design is to select the key geometric factors affecting on its performance as possible, because the computational consumption is highly influenced by the number of the geometric parameters. Many investigations were conducted on the optimization design by changing the individual geometric parameters affecting on the performance of the draft tubes, but these methods were not flexible for improving the performance of the draft tube. Commonly, the elbow draft tube has three parts such as the discharge cone, elbow and exit diffuser. Figure 1 shows the meridian section and the exit cross section of the elbow draft tube.

Figure 1. Geometric parameters of the elbow draft tube

The change of its meridian section affects highly on the performance of the draft tube. Here, the inlet diameter of the draft tube (d_3) is fixed as constant, because its dimension is determined by the runner geometry. The shape of the discharge cone in the draft tube is significantly affected on its hydraulic performance. Generally, the complexity of the flow leaving from the runner outlet is produced by non-uniform distribution of the velocity, which is operated with the certain swirling flow. The discharge cone part of the draft tube gives rise to the secondary flow such as the flow separation from the wall or reverse flow such as the flow separation from the wall or reverse flow in the central region, hence the discharge cone angle α in the optimization of the draft tube should be considered. While the

increment at the height (h_1) of the discharge cone helps to obtain a uniform flow leaving from the runner outlet, but it results in the increase of the hydraulic loss because of the decrease in the curvature radius of the elbow portion if height (h_1) is larger. Therefore, how to set up the rational ration of h_1 and h_2 is very important. The discharge cone part can be featured either by the divergence angle α and cone height h_1 or by cone height h_1 and diameter d_4, the cone height h_1 and diameter d_4 used in this paper. And the curvature radius (r_1), (r_2) and elbow height (h_2) of the draft tube describe the geometrical range of the elbow part through the change of their sizes. The reasonable size of height (h_3) can be decreased the hydraulic losses from the elbow-induced secondary flows, and the height (h_3) of exit diffuser is an important geometrical parameter for the discharge balance between the elbow portion and exit diffuser. The purpose of the exit diffuser is to connect the elbow portion and the tailrace for achieving the higher recovery of the pressure from flow downstream in an elbow. Therefore, the exit diffuser should be considered to minimize the flow separation and reverse flow. Flow separation and reverse flow in the exit diffuser can be effectively controlled by parameters L, L_2, L_3, h_4, θ, and h_5. Because L is constant while the diffuser angle θ can be described as a function of h_4 and L_3, L_3 can be described as a function of the length L. Therefore, the exit diffuser can be indicated by the parameters L, L_2 and h_4. Combining the median section and cross-section is reasonable in the optimization the performance of the draft tube, because the cross-sections of draft tube affect on its performance. Thus, the free geometric parameters for optimization design of the draft tube are nine parameters: h_1, h_3, h_4, d_4, L_1, L_2, r_1, r_2 and w.

2.2 Objective function

The choice of the objective function for the optimization design is very important for the successful solution. In general, the pressure recovery factor (C_p)of the draft tube is the objective function for its performance evaluation. It is desirable that the pressure recovery factor C_p is great as possible in the optimization process. The pressure recovery factor (C_p)indicates the degree of converting of the kinetic energy to the static pressure where a higher value means higher efficiency for the draft tube. The pressure recovery factor is defined as follows:

$$C_p = \frac{p_{out} - p_{in}}{\frac{1}{2}\rho(\frac{Q_{in}}{A_{in}})^2} \tag{1}$$

where p_{in} is the static average pressure at the inlet of the draft tube, p_{out} is the static average pressure at the outlet, A_{in} is the cross-section area of the inlet, Q_{in} is the flow rate at the inlet and ρ is the water density.

The optimization problem of draft tube is mathematically formulated as follows:

$$\begin{cases} maximize: f(X) = C_p \\ over: X = (h_1,\ h_3,\ h_4,\ d_4\ L_1,\ L_2,\ r_1,\ r_2, w) \\ \text{subject to: } X_{lb} \leq X \leq X_{ub} \end{cases} \tag{2}$$

where X is the set of the geometrical parameters of draft tube, *f(X)* is the objective functions for the pressure recovery factor C_p, X_{lb} and X_{ub} is the lower and the upper bounds of the design space for each geometrical parameter.

3. Numerical procedure

3.1. Modeling and meshing.

This step is to build 3D model of the flow domain in the draft tube. The important problem for building the 3D model of the draft tube is to find an adequate method, so as to be easy for modifying its geometrical parameters, because its 3D model is continually refined by changing the free parameters during the automatic optimization process. Hence, Q Xueyi et al. [8] proposed a new method for overcoming the defect of the complex method for the 3D model of the draft tube, which was simple, generating the smooth surface with a large adaptability. Using this method, the 3D model of the draft tube is generated using GAMBIT software in this paper. GAMBIT software automatically

can record the creation process of the 3D model using journal file, which can be modified combining with ISIGHT software in optimization process. While, the computation domain contains the objects such as spiral case, 16 stay vanes, 24 guide vanes, runner with 19 blades and draft tube. Here, in order to improve the accuracy of CFD numerical simulation, the computational domain of the draft tube is divided into three regions such as discharge cone, elbow and exit diffuser. The mesh of the cone portion and exit diffuser portion are comprised of the hexahedral elements, the elbow portion is adopted on the unstructured tetrahedral elements because of the complexity in the flow passage. Then it is generated the entire 3D model of the hydraulic turbine by combining draft tube with the other parts of the hydraulic turbine (spiral casing, stay vane, guide vane, runner) (Figure 2). It is only changed the geometric shape and size of the draft tube; the other parts are not changed in the optimization process. For consideration of the mesh independency, six cases with different number selected between 2 million and 8 million are considered. According to the result, when the mesh number is reached more than 7 million, the efficiency correlation coefficient is 0.05%, therefore the influence of mesh number can be ignored. As a result, the mesh number of 7 million during the optimization solution is selected.
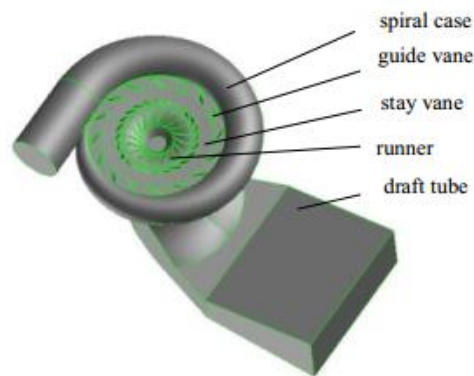
**Figure 2.** 3D model of Francis turbine

3.2. Numerical simulation.
ANSYS FLUENT16.1 is utilized to investigate the flow field of the draft tube, which solves 3D Reynold-Averaged Navier-Stokes equations with the steady state. And Shear Stress Transport (SST) k-ω turbulence model proposed by F R Menter [9] is applied for the turbulence treatment. The SIMPLEC algorithm is adopted to realize the pressure-velocity coupling. The SST model is suggested for the high accuracy of the boundary layer simulation, which can give a more reliable simulation result for flows around complex objects, flows with inverse pressure gradient, transonic flows, etc. The inlet boundary condition is specified as the total pressure at the spiral case inlet and the pressure outlet is specified at the draft tube outlet. The non-slip wall boundary condition is adopted. The standard discretization scheme is employed for the pressure, while the momentum, turbulence kinetic energy and turbulence dissipation rate are used as the second-order upwind scheme. The under-relaxation factor is applied as the default value.

4. Design of experiment
In design and development of the previous product, it has been performed through the typical designing cycle of the design-assessment-redesign with changes of the geometrical design parameters for obtaining of the final design scheme, which includes the design process, numerical simulation and analysis. These processes are calculated repeatedly until they satisfy the condition of the problem, but it needs the consumption of many times due to the numerical simulation and result analysis for each model. And the previous design is usually based on experience or tests. For overcoming of this difficulty of the manual process, we integrate the processes such as the variable 3D design, the numerical simulation and the result analysis using ISIGHT platform. This integration process not only reduces the design cycle time, but also performs the result analysis automatically.

In this paper, DOE analysis of the draft tube by optimization design method is built by using ISIGHT platform of the multi-disciplinary collaborative design. The choice of the sample points for the engineering optimization problems is very important. The DOE technique includes the factorial, central composite, Box-Behnken and face centered composite design, which can solve the optimal problem and conduct the sensitivity analysis for the design space. The optimal Latin hypercube (OLH) is a modified Latin Hypercube design, which generate more evenly distributed the sample points than other DOE strategy [10, 11]. Figure 3 (a) shows a random Latin hypercube. This matrix includes nine design points for two factors and there are nine levels for each factor, permitting higher order polynomial models to be fit to the data and greater assessment of nonlinearity. But, the design points are not distributed evenly in the design space. Figure 3 (b) shows an OLH design matrix. The OLH method can optimize the order of the levels in each column to obtain the combinations of the design points distributed evenly within the design space. The OLH method provides the regular sample points between the lower and upper bounds. In this paper, 60 sets of the sample points are generated using the OLH method in the DOE technique. Here, the range of design variable is [-10%, 10%], while the constraint conditions of the each geometrical parameters is described in Table 1.

(a) Random Latin hypercube   (b) Optimal Latin hypercube
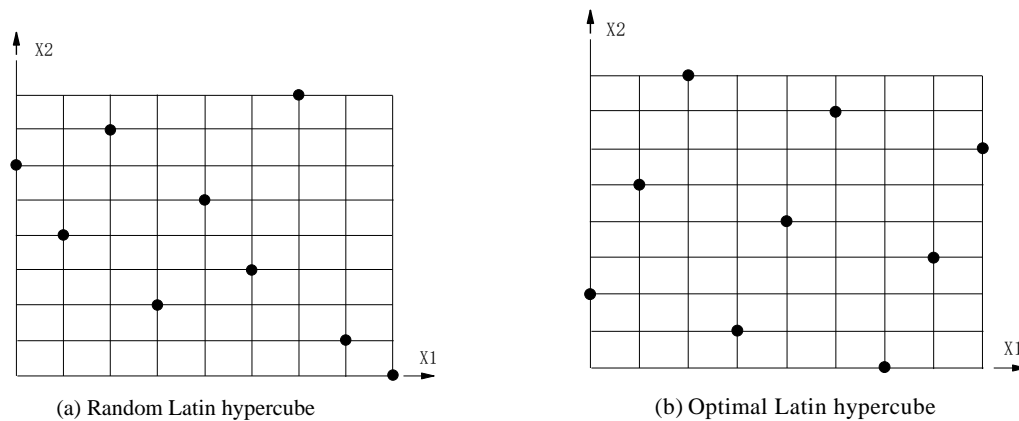
Figure 3. Latin hypercube design and optimal Latin hypercube design

Table 1. Design variable and constraint values at initial design

Variable	Lower Bound	Initial design	Upper bound
h_1	352.700	391.88	431.07
h_3	594.020	660.01	726.01
h_4	-186.610	-169.65	-152.68
d_4	171.840	190.93	210.02
L_1	379.730	421.91	464.09
L_2	85.867	95.41	104.95
r_1	211.110	234.56	258.03
r_2	336.340	373.71	411.08
w	358.040	397.82	437.60

5. Results and discussion

After the completion of the calculation for the generated sampling points according to OLH method, DOE module analyzes the effect of the key geometrical parameters on the pressure recovery factor of the draft tube. The sensitivity relationship between the various geometric parameters of the draft tube is studied at the same boundary conditions. Figure 4 shows the Pareto chart of the design variable effects on C_p.
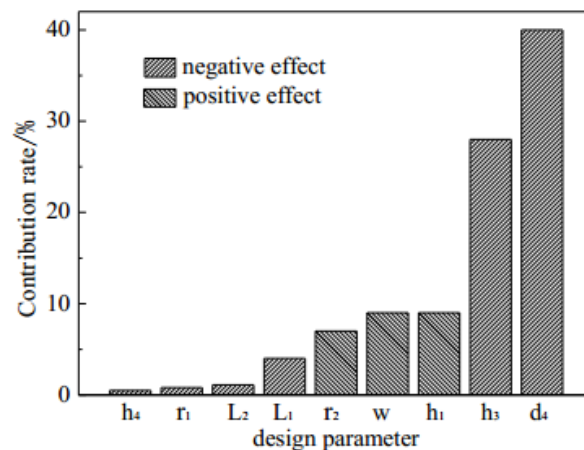
Figure 4. Pareto chart for the pressure recovery factor C_p

The Pareto chart reflects the degree of influence of the geometrical design variable on the performance of draft tube. As shown in the figure 4, h_1, w, r_2 and h_4 have the positive effects on C_p, but d_4, h_3, L_1, L_2 and r_1 have the negative effects. From the ordered bar chart, we can know that the geometrical design parameters (d_4, h_3, h_1, w and r_2) of the draft tube have the higher weight value on its performance. Then the contribution rate of the key geometrical parameters are 40%、28%、9%、7% and 4.5%, respectively. It can be explained that the divergence angle of the inlet cone section, the outlet height of the elbow section, the height of the inlet cone section, the width of the exit diffuse section and the curvature radius of the elbow section are the main geometrical parameters affecting the performance of draft tube. Table 2 gives the optimal geometrical parameters of the draft tube obtained using the response surface method.

Table 2. Optimization results

| Variable	Initial design	Optimal value
h_1	391.88	397.80
h_3	660.01	624.70
h_4	-169.65	-176.67
d_4	190.93	173.77
L_1	421.91	383.10
L_2	95.41	92.60
r_1	234.56	251.00
r_2	373.71	375.00
w	397.82	364.50

The result comparison of the objective functions for the initial and optimal draft tube are listed in Table 3.

Table 3. The objective functions for initial and optimal draft tube

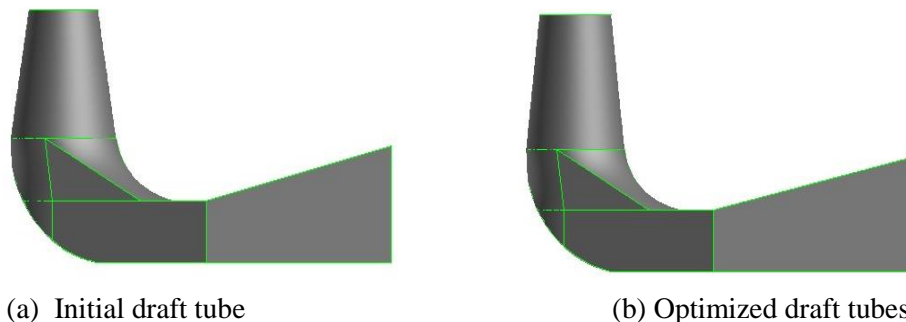| Objective function	Initial design	Optimal value
C_p	0.75	0.8

As shown in Table 3, the results indicate that C_p increases from 0.75 to 0.8
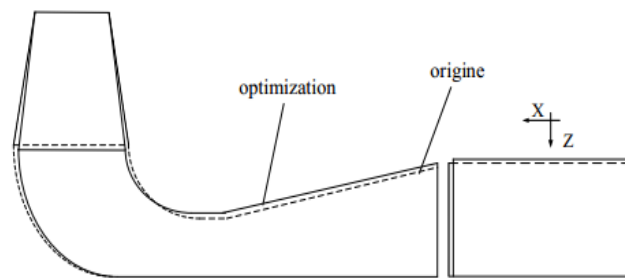
Figure 5. The shape comparison before and after optimization of draft tube

Figure 5 and Figure 6 shows 2D and 3D model of the original and optimized model of the draft tube, respectively.

(a) Initial draft tube   (b) Optimized draft tubes
Figure 6. Comparison of 3D model of draft tube

6. Conclusion
In this paper, we have analyzed the influence of the geometrical parameters that affect on performance of the draft tube and improved its performance. The process has performed using the combination of GAMBIT, ANSYS FLUENT and DOE based on ISIGHT platform. It has been built the 3D model using the GAMBIT software and carried out the performance calculations and analysis of the draft tube using ANSYS FLUENT software. This integration process not only reduces the design cycle time, but also perform the result analysis automatically. Through the optimization process using the combination on CFD and DOE, C_p increased from 0.75 to 0.8, therefore the suggested methodology can be effectively applied for the performance improvement of the hydraulic turbine in the future.

7. Reference
[1]   Prasad V, Khare R and Chincholikar A 2010 Hydraulic performance of elbow draft tube for different geometric configurations using CFD *IGHEM-2010, Ahec, IIT, Roorkee, India, October,*.
[2]   Soni V, Roghelia A, Desai J and et al. 2010 Design development of optimum draft tube for high head Francis turbine using CFD *4th Int. Conf. on Fluid Mechanics and Fluid Power* (Chenni: IIT Madras) pp 16-8.
[3]   Khare R, Prasad V and Verma M 2012 Design Optimisation of conical draft tube of hydraulic turbine *Int. J. of Advances in Engi. Sci. and Technol.* **2** pp 22-6.
[4]   Marjavaara B D and Lundström T S 2006 Redesign of a sharp heel draft tube by a validated CFD-optimization *Int. J. for Numerical Methods in Fluids* **50** pp 911-24.
[5]   Eisinger R and Ruprecht A 2002 Automatic shape optimization of hydro turbine components based on CFD *TASK Q,* **6** pp 101-11.
[6]   Puente L, Reggio M and Guibault F 2003 Automatic shape optimization of a hydraulic turbine draft tube *Int. Conf. CFD2003* (Vancouver: The CFD Society of Canada) p 28.
[7]   Ruprecht A, Eisinger R and Göde, E 2000 Innovative Design Environments for Hydro Turbine Components *HYDRO, Bern* pp 1-10.

[8]   Qi Xueyi, Ma Huiping, Yang Guolai and et al. 2009 3-D Modeling of hydraulic turbine draft t ube based on Pro/E *Transactions of the Chinese Society for Agricultural Machinery* **40** pp 103-6.

[9]   Menter F R 1994 Two-equation eddy-viscosity turbulence models for engineering applications *AIAA J.* **32** pp 1598-605.

[10]   Liefvendahl M and Stocki R 2006 A study on algorithms for optimization of Latin hypercubes *J. of Statistical Planning & Inference* **136** pp 3231-47.

[11]   Zhao M and Cui W C 2007 Application of the optimal Latin hypercube design and radial basis function network to collaborative optimization *J. of Marine Sci. & Appl.* **6** pp 24-32.