Received January 2005
Revised February 2006
Accepted February 2006

# Response surface-based shape optimization of a Francis draft tube

Daniel Marjavaara and Staffan Lundström
*Division of Fluid Mechanics, Luleå University of Technology, Luleå, Sweden*

## Abstract

**Purpose** – This paper aims to develop an efficient and accurate numerical method that can be used in the design process of the waterways in a hydropower plant.

**Design/methodology/approach** – A range of recently published (2002-2006) works, which aim to form the basis of a shape optimization tool for flow design and to increase the knowledge within the field of computational fluid dynamics (CFD) and surrogate-based optimization techniques.

**Findings** – Provides information about how crude the optimization method can be regarding, for example, the design variables, the numerical noise and the multi objectives, etc.

**Research limitations/implications** – It does not give a detailed interpretation of the flow behaviour due to the lack of validation data.

**Practical implications** – A very useful flow design methodology that can be used in both academy and industry.

**Originality/value** – Shape optimization of hydraulic turbine draft tubes with aid of CFD and numerical optimization techniques has not been performed until recently due to the high CPU requirements on CFD simulations. The paper investigates the possibilities of using the global optimization algorithm response surface methodology in the design process of a full scale hydraulic turbine draft tube.

**Keywords** Optimization techniques, Fluid dynamics, Flow, Water power

**Paper type** Research paper

## Introduction

In the present energy market, the demands on rehabilitation and modernisation of old hydropower constructions are increasing due to the fact that a great number of plants are ageing and that they are run at off-design conditions. Today, however, often only the generator and/or the runner are considered when refurbishing a system, although several investigations show the importance of the waterways (Avellan, 2000; Scottas and Ryhming, 1993; Turbine-99, 1999). For example, the efficiency of a low or medium head hydraulic turbine is significantly affected by the performance of its draft tube (Gubin, 1973; Krivchenko, 1994).

Traditionally, the design of hydraulic turbine draft tubes has been based on model tests and the experience of the designing engineers. With the latest advances in computer performance, computational fluid dynamics (CFD) has also matured as a useful design tool in the search for optimal solutions regarding, for instance,

The authors gratefully acknowledge the support provided by Vattenfall Utveckling AB, the Swedish Agency for Innovation Systems (VINNOVA) and the participating industries in the Polhem Laboratory.



0961-5539
DOI 10.1108/09615530710716072

performance, flow behaviour, construction consideration and/or economical aspects (Avellan, 2000; Scottas and Ryhming, 1993; Turbine-99, 1999). Shape optimization of hydraulic turbine draft tubes with aid of CFD and computer optimization has, however, not been performed until recently due to high CPU requirements on 3D CFD calculations (Eisinger and Ruprecht, 2001; Marjavaara and Lundström, 2003; Marjavaara and Lundström, 2006). Such optimizations methodologies have nevertheless shown on the potential of using CFD and computer optimizations in the design/rehabilitation process of a hydraulic turbine draft tube. Several problems have though to be solved before it can routinely be applied in product development. One problem is that the stability of the optimum found is often unknown and that it is not evident how to choose the best optimization technique. Another is that the accuracy and reliability of the CFD predictions is highly dependent on the boundary conditions and the grid (Cervantes, 2003; Engström, 2003; Mauri, 2002; Turbine-99, 1999). Conversely, there are many benefits of a fully working procedure. It will, for instance, be possible to get reliable results in reasonable time, to find new products with better functionality and to save cost and time in product development, maintenance and support. It is, therefore, vital to further develop, verify and validate both CFD calculations and computer optimization techniques.

In the present study, the response surface methodology (RSM) will be investigated in order to optimize the shape of a Francis draft tube (Myers and Montgomery, 2002). This optimization strategy is very common in other engineering applications with demanding, complex and expensive numerical simulations such as CFD calculations (Madsen *et al.*, 2000; Shyy *et al.*, 2000). The main advantages with the RSM, compared to most other optimization techniques, are that it is robust and that it is efficiently filter noise intrinsic to numerical or experimental data (Beaton and Tuckey, 1974; Burman and Gebart, 2001). Additional advantages are its strong coupling to design variable and design space characteristics, and its parallel computing capabilities. The disadvantages are that the computer requirements grow very fast with increasing number of design variables.

One way to improve the quality of a response surface (RS) and minimise the effect of noise is to use a design of experiment (DOE) strategy when selecting data points for the construction of it (Myers and Montgomery, 2002). Another way is to identify and repair data points that have too strong influence on the fidelity of the RS, so-called outliers. Here, the face centred composite design (FCCD), respectively, the iterative re-weighted least square (IRLS) method is adopted for these purposes (Beaton and Tuckey, 1974; Myers and Montgomery, 2002). The overall objective is to minimise the draft tube losses by maximising the average pressure recovery factor and minimising the energy loss factor, one by one and simultaneously. Issues to be addressed are numerical noise due to discretization, iterative, round off and modelling errors, and trade-offs caused by multiple objectives.

The first three sections describe the assumptions and procedure of the CFD simulations and the RS-based optimization. The result is then presented and discussed in the rest of the paper.

## Geometric description

The flow domain is a full-scale model of a curved draft tube, see Figure 1, built and installed 1963 at Yngeredsforsen hydropower plant in Sweden. The inlet and bend part (cone, respectively, elbow) of the draft tube geometry is a steel construction, while the

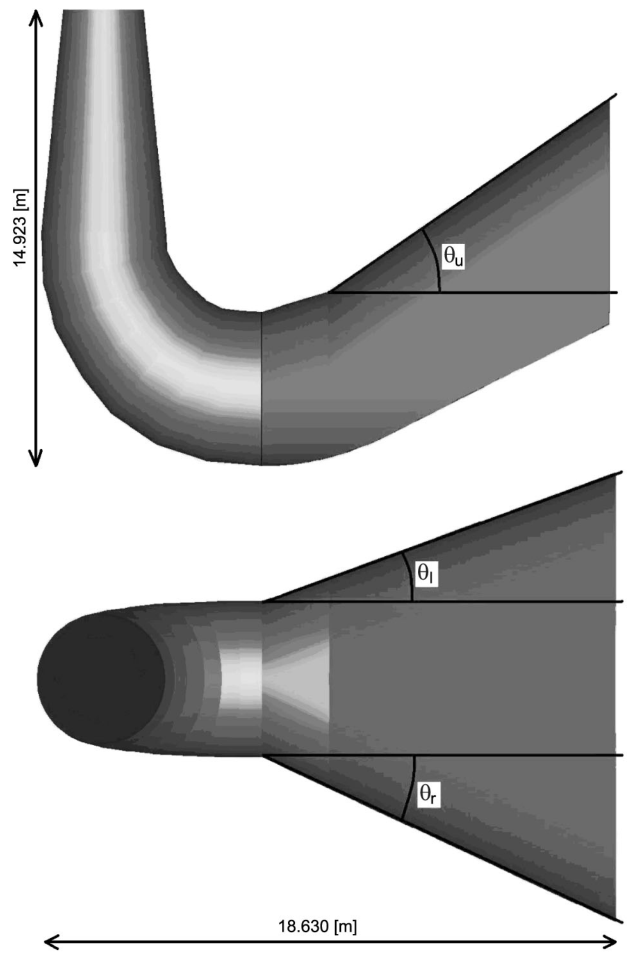


**Figure 1.**
The draft tube geometry and its design variables

end diffuser is a concrete construction. Therefore, it is expected, from a construction point of view, that the end diffuser is both easier and less expensive to rebuild, although it is likely that the largest efficiency improvements are found when the cone is optimized. Hence, the design variables considered here are, the right end diffuser wall angle $\theta_r$, the left end diffuser wall angle $\theta_l$ and the upper end diffuser wall angle $\theta_u$, as shown in Figure 1. By alterations in the end diffuser geometry it is also anticipated that the velocity field becomes more uniform after the elbow and thereby the distortion and the overall losses are reduced. The actual parameterization of the draft tube geometry is done in accordance with the Adapted design method presented in Marjavaara and Lundström (2003).

The main purpose with the draft tube is to utilize the energy corresponding to the hydraulic turbine installation height above the tail water level and the water kinetic energy at the runner outlet. Typical measures of its performance are therefore the pressure recovery factor and the energy loss factor, which both are highly influenced by the operating point of the turbine. Although the objective function could contain other parameters involving economical aspects, for instance, these two measures will be used here in the RS-based optimization process. The average pressure recovery factor $C_p$ is defined as:

$$C_p = \frac{\frac{1}{A_{out}} \int_{A_{out}} \int p \, dA - \frac{1}{A_{in}} \int_{A_{in}} \int p \, dA}{\frac{1}{2} \rho \left( \frac{Q_{in}}{A_{in}} \right)^2} \tag{1}$$

where $A$ is the area perpendicular to the main flow direction, $p$ the static pressure, $Q$ the flow rate, $\rho$ the density and the subscripts in and out corresponds to inlet and outlet, respectively. The energy loss factor $\zeta$ in its turn is defined as:

$$\zeta = \frac{\int_{A_{in}} \int p_{tot} \mathbf{u} \cdot \mathbf{n} \, dA + \int_{A_{out}} \int p_{tot} \mathbf{u} \cdot \mathbf{n} \, dA}{\int_{A_{in}} \int p_{dyn} \mathbf{u} \cdot \mathbf{n} \, dA} \tag{2}$$

where $\mathbf{n}$ is the surface normal vector, $p_{dyn}$ is the dynamic pressure, $p_{tot}$ is the total pressure and $\mathbf{u}$ is the velocity vector

**Optimization approach**

The RSM approach implies that a series of experiments is evaluated, numerically or experimentally, for a prescribed set of data points. The sampled response (objective function) is then used to construct a global approximation, called RS, of the performance over the design space. An optimal design can then finally be found at low costs from the RS because it is merely an algebraic expression. Depending on the behaviour of the RS near the optimum one can decide if the approximation has to be refined or not. Common quality statistics for this purpose are the coefficient of multiple regression $R^2$ and its adjusted form $R_a^2$. Other relevant quantities are the root mean square error $\sigma$, and its revised counterpart $\sigma_a$.

Here, quadratic RS models are generated using the statistical analysis software JMP (2002), since the number of data points is rather few. This fitting procedure is actually a least square problem so it is computationally straightforward. The prescribed data points are selected according to the FCCD DOE technique, in order to get an optimal number and distribution of the data points. It generates $(2^n + 2n + 1)$ data points, where $n$ is the number of design variables located at the centre, corners and face centres of the design space. The optimal designs are finally found with the generalised reduced gradient algorithm in Microsoft Excel Toolbox (JMP, 2002; Ladson *et al.*, 1978).

The fidelity of the RS models is improved by usage of the IRLS approach. It detects and eliminates data points (outliers) that have too strong influence on the statistics and thereby reduce the quality of the RSs. In principal low weights are assigned to these outliers and by refitting the RSs with a weighted least square procedure the effects of the outliners can be suppressed. This process is finally repeated until convergence. The weight $w$ that is given to a data point in this case is:

$$w_i = \begin{cases} \left[ 1 - \left( \frac{|e_i/\sigma_a|}{B} \right)^2 \right]^2, & \text{if } |e_i/\sigma_a| \leq B, \\ 0, & \text{otherwise,} \end{cases} \tag{3}$$

where $B$ is a tuning parameter usually having a value between 1 and 3, and $e$ is the residual. In this case a value of 1.9 was chosen for $B$.

The multi-objective optimization problem can be summarised as follows:

$$\begin{aligned} \text{Maximise}: \; & f = (C_\mathrm{p}, -\zeta, D)^\mathrm{T} \\ \text{Subject to}: \; & g = (\theta_\mathrm{r} + \theta_\mathrm{l})^\mathrm{T} \geq 0, \\ & -45.0° \leq \theta_\mathrm{r} \leq 45.0°, \\ & -45.0° \leq \theta_\mathrm{l} \leq 45.0°, \\ & 25.2° \leq \theta_\mathrm{u} \leq 39.2°. \end{aligned} \tag{4}$$

where $f$ contains the objective functions, $g$ contains the geometrical inequality constrains, and the remaining side constraints are the bounds. The inequality constraint is introduced in order to ensure that the draft tube outlet is wider or equal to the original outlet, and the side constraints are defined to avoid model and grid generation difficulties. The three objective functions, which are optimized one by one, are the average pressure recovery factor $C_\mathrm{p}$, the energy loss factor $\zeta$ and the composite desirability function $D$. The latter optimizes $C_\mathrm{p}$ and $\zeta$ simultaneously, by relating the target values to each other (Shyy *et al.*, 2000). This technique is often applied in multi-objective optimization problems, and it builds up a composite RS from individual RSs by using a geometric mean. The resulting function that will be maximised becomes:

$$D = \sqrt{d_{C_\mathrm{p}} \cdot d_\zeta} \tag{5}$$

where the individual desirability functions of $C_\mathrm{p}$ and $\zeta$, $d_{C\mathrm{p}}$, respectively, $d_\zeta$, are defined as:

$$d_{C_\mathrm{p}} = \left(\frac{C_\mathrm{p} - L}{T - L}\right)^\mathrm{r} \tag{6}$$

respectively,

$$d_\zeta = \left(\frac{\zeta - H}{T - H}\right)^\mathrm{r} \tag{7}$$

where $H$ is the highest acceptable value, $L$ is the lowest acceptable value, $T$ is the target value and the powers $r$ set the rule of each individual desirability function (here set equal to one).

With three design variables, the standard FCCD technique yields 15 data points. However, in this case, four of these points violate the inequality constraint given in equation (4). In order to improve the final DOE population, seven additional data points that are satisfying the inequality constraint are added. The total number of data points in the design space then becomes 18, as shown in Figure. 2. The values of the design variables for the original draft tube are $\theta_\mathrm{r} = 0°$ $\theta_\mathrm{l} = 0°$ and $\theta_\mathrm{u} = 32.8°$.

## CFD simulations

The assumed steady 3D turbulent flow field, in each draft tube geometry, is solved with the CFD code CFX-5 (CFX-5, 2004). This code has a coupled solver, and utilizes

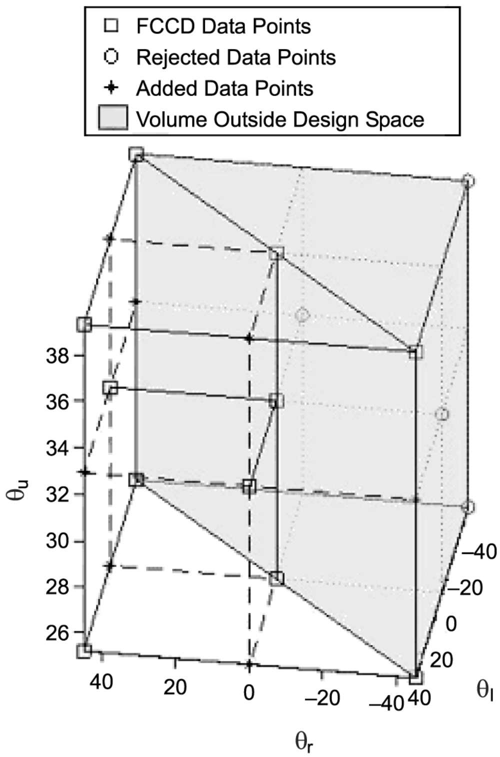


**Figure 2.**
The modified FCCD population for the construction of individual RSs

a finite element based finite volume method applied on an unstructured grid to solve the governing equations. In this work, the discretization of the pressure gradient term and the diffusion term are obtained with shape functions, while the discretization of the advection term is done with the high resolution scheme (HRS). The HRS evaluates automatically and locally a blend factor $\beta$, which determines the level of correction for the upwind difference scheme (UDS), as close to one without violating the boundedness principles. A value of zero corresponds to the first order UDS scheme and a value of one to a second order accurate scheme.

The turbulence is modelled with the standard $k$-$\varepsilon$ model and by usage of scalable wall functions (CFX-5, 2004). Although it is well known that this model is unable to reproduce all details of the flow accurately (Durbin and Reif, 2000), it is chosen since there are no major reported improvements for alternative models regarding draft tube flow simulations (Turbine-99, 1999). It is also used here so a general optimization framework first can be established and general findings be investigated. For the discretization of the turbulence equations, first order UDS scheme are employed in order to get a robust and well converged solution.

In the absent of authentic boundary conditions, the inlet boundary components are taken from measurements of the best efficiency operating point of the GAMM Francis hydraulic turbine (Nilsson, 2002). This Francis model, including stay vanes, guide vanes, runner and draft tube, was used as a test case at the GAMM Workshop on 3D Computation of Incompressible Internal Flows 1989 (Scottas and Ryhming, 1993). The velocity components are scaled and adjusted to fit the dimension of the present draft tube, and to get the required flow rate of 38 m/s. For the turbulent quantities, a turbulence intensity of 5 percent and a turbulence length scale of 10 percent of the inlet diameter are assumed. At the outlet an average zero total pressure boundary condition is applied, which allows both flow into and out of the domain. To ensure the validity of this boundary condition, the outlet of the draft tube geometry is also extended 10.0 m downstream. Finally, the walls are assumed smooth with a no-slip boundary condition applied.

The draft tube geometry is built and modified with the CAD software I-DEAS according to the geometric parameterisation previously discussed (I-DEAS, 2003). The grids, containing wedges elements near the walls and tetrahedral elements in the bulk flow, are generated with the grid generator ICEM CFD (2004). To minimise the numerical errors and the influences from grid topologies, the same mesh parameters are used for all draft tube geometries, such that the grid is only modified in the end diffuser. Grids of different layout were also tested in order to determine the best one in terms of convergence and execution time. The selected grid configuration consisted of 1.1M elements, and has a minimum angle of 12° and an average aspect ratio of 110.

## Results

Totally, 21 CFD calculations were conducted for the construction and validation of the RS approximations. All of these simulations are presented in Table I, where the first

**Table I.**
Result from the CFD analysis

| Count | $\theta_r$[°] | $\theta_l$[°] | $\theta_u$[°] | $C_p$ | $\zeta$ | $D$ |
|---|---|---|---|---|---|---|
| *DOE designs* | | | | | | |
| 1 | −45 | 45 | 25.2 | 0.832 | 0.144 | 0.844 |
| 2 | −45 | 45 | 32.8 | 0.886 | 0.142 | 0.872 |
| 3 | −45 | 45 | 39.2 | 0.906 | 0.143 | 0.881 |
| 4 | 0 | 0 | 25.2 | 0.895 | 0.116 | 0.890 |
| *5*[a] | *0* | *0* | *32.8* | *0.911* | *0.117* | *0.897* |
| 6 | 0 | 0 | 39.2 | 0.912 | 0.119 | 0.897 |
| 7 | 0 | 45 | 25.2 | 0.907 | 0.142 | 0.882 |
| 8 | 0 | 45 | 32.8 | 0.901 | 0.143 | 0.879 |
| 9 | 0 | 45 | 39.2 | 0.895 | 0.144 | 0.875 |
| 10 | 45 | −45 | 25.2 | 0.852 | 0.119 | 0.867 |
| 11 | 45 | −45 | 32.8 | 0.902 | 0.119 | 0.891 |
| 12 | 45 | −45 | 39.2 | 0.918 | 0.120 | 0.899 |
| 13 | 45 | 0 | 25.2 | – | – | – |
| 14 | 45 | 0 | 32.8 | 0.902 | 0.141 | 0.881 |
| 15 | 45 | 0 | 39.2 | 0.897 | 0.142 | 0.877 |
| 16 | 45 | 45 | 25.2 | – | – | – |
| 17 | 45 | 45 | 32.8 | 0.895 | 0.161 | 0.866 |
| 18 | 45 | 45 | 39.2 | 0.893 | 0.160 | 0.866 |
| *Validated optimal designs (RS models)* | | | | | | |
| 19 | 7.9 | −7.9 | 39.2 | *0.912* | 0.117 | 0.897 |
| 20 | 19.6 | −19.6 | 25.2 | 0.892 | *0.113* | 0.890 |
| 21 | 12 | −12 | 39.2 | 0.912 | 0.117 | *0.898* |

**Note:** [a]Original design

18 correspond to the DOE data points and the last 3 to the CFD validated optimal solutions. The convergence criteria was set to a residual drop of at least three decades, for all MAX and RMS residuals, which is sufficient for most industrial applications according to (Casey and Wintergerste, 2000). However, 2 of the 18 DOE configurations (count 13 and 16 in Table I) did not fulfil this criteria, so they were rejected from the analysis. The average $y^+$ value for all calculations at near wall nodes was 190 (range from 1 to 580), which is not optimal based on established practices (Casey and Wintergerste, 2000). It is, however, still good for this complex flow situation and compares to other similar works if the high Reynolds number is accounted for as well (Turbine-99, 1999).

For the construction of the quadratic RS approximations, three models were generated for each objective (i.e. $C_p$, $\zeta$ and $D$), as summarized in Tables II and III:

(1) RS model based on the original DOE population;

(2) IRLS model based on the same population; and

(3) IRLS models based on an enhanced population.

| Objective | $R^2$ | $R_a^2$ | $\sigma$ | Mean | Observations |
|---|---|---|---|---|---|
| *RS models based on the original DOE population* | | | | | |
| $C_p$ | 0.802 | 0.504 | 0.016 | 0.894 | 16 |
| $\zeta$ | 0.998 | 0.996 | 0.001 | 0.136 | 16 |
| $D$ | 0.868 | 0.671 | 0.008 | 0.879 | 16 |
| *IRLS models based on the original DOE population* | | | | | |
| $C_p$ | 0.818 | 0.490 | 0.006 | 0.901 | 8.7 |
| $\zeta$ | 0.999 | 0.999 | 0.001 | 0.132 | 8.5 |
| $D$ | 0.950 | 0.860 | 0.003 | 0.883 | 8.7 |
| *IRLS models based on the enhanced DOE population* | | | | | |
| $C_p$ | 0.963 | 0.921 | 0.002 | 0.903 | 13.2 |
| $\zeta$ | 0.999 | 0.999 | 0.001 | 0.125 | 12.4 |
| $D$ | 0.992 | 0.983 | 0.001 | 0.887 | 13.1 |

**Table II.** Properties of the RS and IRLS models

| Objective | $\theta_r$[°] | $\theta_l$[°] | $\theta_u$[°] |
|---|---|---|---|
| *RS models based on the original DOE population* | | | |
| $C_p$ | 7.9 | −7.9 | 39.2 |
| $\zeta$ | 19.6 | −19.6 | 25.2 |
| $D$ | 12.0 | −12.0 | 39.2 |
| *IRLS models based on the original DOE population* | | | |
| $C_p$ | 13.6 | −13.6 | 39.2 |
| $\zeta$ | 18.6 | −18.6 | 25.2 |
| $D$ | 15.9 | −15.9 | 39.2 |
| *IRLS models based on the enhanced DOE population* | | | |
| $C_p$ | 17.2 | −17.2 | 38 |
| $\zeta$ | 19.1 | −19.1 | 25.2 |
| $D$ | 16.6 | −16.6 | 36.7 |

**Table III.** Estimated optimal designs from the RSM analysis

In this latter enhanced DOE population, the 3 CFD validated optimal solutions from the first RS models were added to the population and all outlier with a weight less than 0.1 were removed. The original DOE population consisted of the 16 initial data configurations in Table I.

The low $R_a^2$ values for $C_p$ and $D$ for the original RS and IRLS models indicate, however, that these fits are of poor accuracy, see Table II. By adding data points to the original DOE population, the quality of the IRLS model for both $C_p$ and $D$ become more satisfactory, as seen in Table II. The fidelity for $\zeta$ are, however, high for all approximations as shown by the $R_a^2$ values close to unity in Table II. This high accuracy is also noticed in the variation of the estimated optimal values for $\theta_l$ and $\theta_r$ between the different models. For $\zeta$ the variation is low (approximately $\pm 0.5°$), while it is considerable larger for $C_p$ and $D$ (approximately $\pm 5.0°$, respectively, $\pm 2.8°$) due to less accuracy, as seen in Table II and III.

An evaluation of the estimated optimal angles in Table III, shows that the optimal draft tube geometry should have a curved end diffuser rather a straight one. Physically this can be traced to the swirling flow motion in the draft tube. The obtained CFD improvements compared to the original draft tube design validates this conclusion, as seen in Table I where the efficiency increases with 0.1 and 3.4 percent for $C_p$ and $\zeta$, respectively. Interestingly, the $\zeta$ objective suggests an end diffuser formed as a contraction rather than an expansion, see the low value of $\theta_u$ in Table III. This is physically correct since the outlet losses are not accounted for in $\zeta$, but the draft tube becomes unefficient. This result clearly exemplifies that $C_p$ is a better measurement of the draft tube performance than $\zeta$.

Comparable designs are finally identified by plotting regions with small deviation from the optimal design in the same design space for the two objectives $C_p$ and $\zeta$, as shown in Figure 3 where the shaded isosurfaces corresponds to 1 percent, respectively, 3 percent deviation from the optimal designs. Hence, an optimal design for compatible design objectives should have a $\theta_u$ value that is larger than the original design but lower than the optimal design estimated from $C_p$. This in accordance with result for the final IRLS composite objective function $D$, as seen in Table III. Another remark in Figure 3 is that $\zeta$ seems to be less sensitive to non optimal designs than $C_p$, which can be traced to the better fidelity of the $\zeta$ approximations in Table II.

## Conclusion

The present work demonstrates the possibilities and difficulties of using the RSM in the design process of a hydraulic turbine draft tube. It among other things exemplifies the potential of using the IRLS method and multi objectives for improving the estimated optimal design. The overall result is that the RSM approach is capable of offering satisfactory results in the design process of hydraulic components. Furthermore, it increases the knowledge of the flow designer, regarding categorizing the design variables that affect the design objectives and in finding design solutions with comparable design objectives.

The outcome of the optimization indicates that the end diffuser of the draft tube geometry should be curved, due to the swirling flow motion in the draft tube. The pressure recovery, however, is relative insensitive to minor changes of the end curvature (improvements of the order 0.1 percent), suggesting that the CFD simulation might not catch all details of the flow accurately due to the applied turbulence model

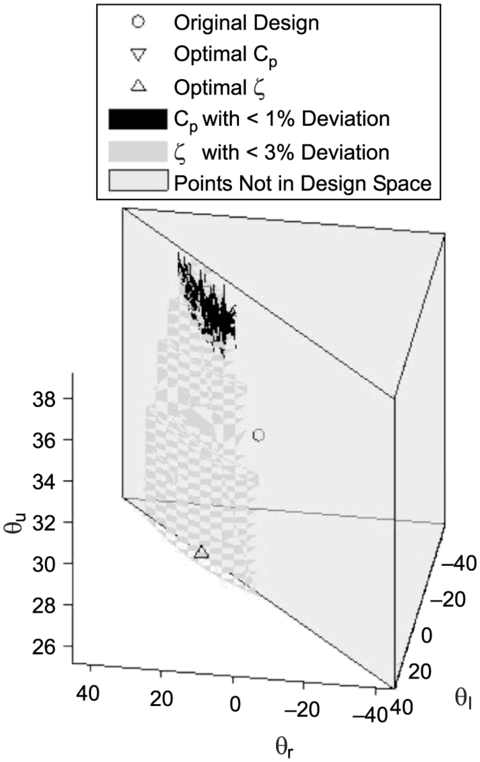


**Figure 3.** Regions in design space with 1 and 3 percent deviation in $C_p$ and $\zeta$, respectively

and/or inlet boundary conditions. Better CFD predictions will, therefore, certainly affect the outcome of the optimization process (magnitude of improvements, optimal values and so on), although the present ones most likely catch the overall trends and primary flow patterns. It should also be kept in mind, that the present CFD calculations were performed in accordance to the best available praxis today regarding draft tube flow simulations (Turbine-99, 1999). Modification of the draft tube geometry could finally also reflect in alterations in the inlet velocity profile, meaning that the runner have to be included in the CFD simulations to get larger improvements in the pressure recovery. The energy loss factor gave better improvements, but should not be used as an objective function for the whole draft tube since the outlet losses are not accounted for in the expression. Furthermore, it is showed that the numerical noise (i.e. the presents of discretization, iterative, round off and modelling errors) can be minimised with adequate result by the usage of an IRLS method. Keeping the size of the design space and modifying the design point distribution in the present case might improve the fits further, since the chosen one was not variance optimal and there were no points in the middle of the design space. The usage of another order of the response surface and/or a reduced design space will certainly reduce the RSM modelling errors and enhance the result.

## References

Avellan, F. (2000), "Flow investigation in a Francis draft tube: the FLINDT project", *Proceedings of 20th IAHR Symposium, Charlotte, North Carolina, USA*.

Beaton, A.E. and Tuckey, J.W. (1974), "The fitting of power series, meaning polynomials, illustrated on spectroscopic data", *Technometrics*, Vol. 16 No. 2, pp. 147-85.

Burman, J. and Gebart, B.R. (2001), "Influence from numerical noise in the objective function for flow design optimization", *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol. 11 No. 1, pp. 6-19.

Casey, M. and Wintergerste, T. (2000), *Special Interest Group on Quality and Trust in Industrial CFD: Best Practice Guidelines*, ERCOFTAC, Version 1.0, pp. 11-19.

Cervantes, M.J. (2003), "Effects of boundary conditions and unsteadiness on draft tube flow", doctoral thesis 2003:11, Luleå University of Technology, Luleå.

CFX® (2004), *CFX-5*, Version 5.7, ANSYS Europe Ltd.

Durbin, P.A. and Reif, B.A. (2000), *Statistical Theory and Modelling for Turbulent Flows*, Wiley, New York, NY.

Eisinger, R. and Ruprecht, A. (2001), "Automatic shape optimization of hydro turbine components based on CFD", *TASK Quarterly*, Vol. 6 No. 1, pp. 101-11.

Engström, T.F. (2003), "Simulations and experiments of turbulent diffuser flow with hydropower applications", Doctoral thesis 2003:10, Luleå University of Technology, Luleå.

Gubin, M.F. (1973), *Draft Tubes of Hydro-electric Stations*, Amerind Publishing Co., New Dehli.

ICEM CFD® (2004), *ICEM CFD*, Version 5.1, ANSYS, Inc.

I-DEAS® (Copyright © 2003), *I-DEAS*, Version 10, Electronic Data System Corporation.

JMP® (2002), *The Statistical Discovery SoftwareTM*, Release 5.0.1a, SAS Institute, Inc.

Krivchenko, G. (1994), *Hydraulic Machines: Turbines and Pumps*, 2nd ed., Lewis Publishers, Boca Raton, FL.

Ladson, L.S., Waren, A.D., Jain, A. and Ratner, M. (1978), "Design and testing of a generalized reduced code for non-linear programming", *ACM Transactions on Mathematical Software*, Vol. 4 No. 1, pp. 34-50.

Madsen, J.I., Shyy, W. and Haftka, R.T. (2000), "Response surface techniques for diffuser shape optimization", *AIAA Journal*, Vol. 38 No. 9, pp. 1512-18.

Marjavaara, B.D. and Lundström, T.S. (2003), "Automatic shape optimization of a hydropower draft tube", *Proceedings of 4th ASME_JSME Joint Fluids Engineering Conference*, Vol. 1, pp. 1819-24.

Marjavaara, B.D. and Lundström, T.S. (2006), "Redesign of a sharp heel draft tube by a validated CFD-optimization", *International Journal for Numerical Methods in Fluids*, Vol. 50 No. 8, pp. 911-24.

Mauri, S. (2002), "Numerical simulation and flow analysis of an elbow diffuser", doctoral thesis, École Politechnique Fédérale de Lausanne, Lausanne.

Myers, R.H. and Montgomery, D.C. (2002), *Response Surface Methodology: Process and Product Optimization Using Designed Experiments*, Wiley, New York, NY.

Nilsson, H. (2002), "Numerical investigation of turbulent flow in water turbines", doctoral thesis, Chalmers University of Technology, Gothenburg.

Scottas, G. and Ryhming, I.L. (1993), "3D-computations of incompressible internal flows", *Proceedings of the GAMM workshop held at EPFL, Notes on Numerical Fluid Mechanics*, Vol. 39, Vieweg Verlag, Braunschweig.

Shyy, W., Papila, N., Tucker, K., Vaidyanathan, R. and Griffin, L. (2000), "Global design optimization for fluid machinery applications", *Proceedings of the Second International Symposium on Fluid Machinery and Fluid Engineering, Beijing, China*.

Turbine-99 (1999), "The IAHR workshops on draft tube flow", available at: www.turbine-99.org (accessed March 2006).

**Corresponding author**
Daniel Marjavaara can be contacted at: dama@ltu.se