DEVELOPMENT, VALIDATION AND APPLICATION OF AN OPTIMIZATION SCHEME FOR IMPELLERS OF CENTRIFUGAL FANS USING CFD-TRAINED METAMODELS

Konrad BAMBERGER[1], Julian BELZ[2]

Thomas CAROLUS[1], Oliver NELLES[2]

[1]University of Siegen, Institute for Fluid- and Thermodynamics
Paul-Bonatz-Strasse 9-11, D-57068 Siegen, Germany

[2] University of Siegen, Institute for Mechanics and Control Engineering,
Paul-Bonatz-Strasse 9-11, D-57068 Siegen, Germany

SUMMARY

A quick and reliable method for the aerodynamic optimization of centrifugal impellers is presented. It is based on an evolutionary optimization algorithm that indentifies the optimal geometrical parameters for a given aerodynamic objective function. The quickness of the method stems from evaluating the objective function using artificial neural networks (ANN). The ANNs were trained with a dataset containing approximately 4,000 characteristic curves obtained from Computational Fluid Dynamics (CFD) wherefore the ANNs are also named "metamodels" of CFD. It is demonstrated in the paper that the newly developed metamodels differ from previous metamodels in terms of universality since they can be used for optimizing all typical design points of centrifugal impellers according to the Cordier diagram.

The new optimization scheme was applied to numerous design points and the resulting geometries were simulated by CFD. It was found that the metamodels tend to overpredict efficiency at design points with untypically large specific fan diameters. At standard design points, however, CFD confirms well the metamodel predictions proving the broad applicability of the optimization scheme. The optimizations in addition yield a map of achievable efficiencies as a function of the design point.

The CFD model was experimentally validated with a total of seven prototypes. The prototypes were selected aiming at maximal geometrical and aerodynamic diversity. Due to the good agreement between CFD and experiment, the optimization scheme is considered to be successfully validated.

INTRODUCTION

Classic design methods for centrifugal fans are for instance described by Pfleiderer [1, 2] and Bommes [3]. The main advantage of these methods is the low demand for computational resources. However, being based to a considerable degree on empiricism, such methods lead to suboptimal efficiencies and accuracies with regard to the fulfillment of the design target. For that reason, modern fan design is usually supported by Computational Fluid Dynamics (CFD). For instance, the designer can apply the classic methods first, analyze the result by CFD and then try to improve the design based on the interpretation of the CFD results. The full exploitation of the potential for improvement, however, can only be achieved by coupling CFD with optimization algorithms. The main drawback of this method is the associated computational cost. Hence, methods were developed to reduce the required amount of CFD simulations. For instance, Ratter et al. [4-6] succeeded to reduce the CFD effort by incorporating pre-knowledge about the optimal position of the stagnation point at the leading edge of a centrifugal fan blade. Moreover, a response surface method was used to further reduce the number of required CFD simulations. The response surface method is one example of CFD-trained metamodels which have the advantage that CFD is only required to generate a dataset with which the metamodels are trained. After that, the metamodels predict the fan performance several orders of magnitude faster than CFD itself.

The present work is also concerned with the development of metamodels and their application for the aerodynamic optimization of centrifugal impellers. The main improvement over previously developed metamodels is their universality since they can be used to optimize centrifugal impellers for all typical design points in the Cordier diagram. The design point is characterized by the flow rate Q and the total-to-total pressure rise $\Delta p_{tt} = p_{t2} - p_{t1}$ where the index "t" means total and the indices "1" and "2" refer to positions upstream and downstream of the impeller, respectively. For the sake of comparability, the design point should rather be expressed by the non-dimensional flow and pressure coefficients φ and ψ_{tt}:

$$\varphi = \frac{Q}{\frac{\pi^2}{4} D^3 N}, \quad \psi_{tt} = \frac{\Delta p_{tt}}{\frac{\pi^2}{2} D_2^2 N^2 \rho} \tag{1, 2}$$

D_2 is the outer impeller diameter, N is the rotational speed and ρ is the fluid density. In many practical applications the kinetic energy of the fluid downstream of the impeller is useless since it dissipates in the surrounding atmosphere. A definition of the pressure rise which considers the kinetic energy at the impeller outlet as loss is the total-to-static pressure rise Δp_{ts}. The corresponding total-to-static pressure coefficient ψ_{ts} is defined analogously to eq. (2). The efficiency of the impeller is the quotient of air power and the power of the shaft driving the fan:

$$\eta = \frac{Q \Delta p}{P_{\text{shaft}}} = \frac{Q \Delta p}{2\pi N T_{\text{shaft}}} \tag{3}$$

T_{shaft} is the torque of the driving shaft. The efficiency can be computed with both, Δp_{ts} and Δp_{tt}, yielding the total-to-static efficiency η_{ts} and the total-to-total efficiency η_{tt}, respectively.

The focus of this work is to demonstrate a computationally cost-effective way to generate metamodels that predict ψ_{ts} and η_{ts} of centrifugal impellers for a set of geometrical parameters. Moreover, the metamodels are coupled with an evolutionary optimization algorithm and the efficiency is optimized for many design points. The optimized geometries are simulated by means of CFD yielding two major findings: Firstly, the CFD-simulated performance is compared to the metamodel prediction to assess the applicability of the metamodels for aerodynamic optimization. Secondly, the resulting efficiencies are analyzed to determine the maximum achievable efficiency as a function of the design point. Eventually, the CFD model is experimentally validated with a total of seven prototypes.

METHODOLOGY

This section describes the methodology for the development of the optimization scheme. At first, the impeller geometry is parameterized yielding the input space of the metamodels. Seven points of this input space (i.e. seven distinct impellers) are selected for experimental validation of the CFD model. The validated CFD model is then used to simulate the characteristic curves of approximately 4,000 impellers selected by Design of Experiment (DoE). The resulting dataset is used to train the metamodels. Several measures to improve the metamodel quality are described.

Impeller parameterization

The main objective of the impeller parameterization is to enable a high level of geometrical flexibility while keeping the number of free geometry parameters to a minimum. For that reason, basic parameters which are also used in classic literature [2, 3] are selected. Those parameters are graphically illustrated in Fig. 1 and listed in Tab. 1. The blades have a circular shape between the inner diameter D_1 and the outer diameter D_2. The bottom disc is perpendicular to the axis of rotation. The shroud is not parallel to the bottom disc but tapers between D_1 and D_2. The innermost part of the shroud has a circular shape with the radius r_s. The outer part of the shroud has a linear shape determined by the two distances b_1 and b_2 from the bottom disc. The circular arc forming the blade has the constant thickness S. This represents a very simple geometry and thus facilitates cost-effective manufacturing techniques and keeps the number of parameters required to describe the blade shape low. The radius and the centerpoint of the circular arc are given via the inlet angle β_{b1} and the outlet angle β_{b2} which are measured between the circumferential direction and a tangent at the blade leading or trailing edge, respectively. Together with the number of blades z, the aforementioned geometrical parameters are sufficient to describe simple impellers. In order to enable more innovative designs, two more parameters are introduced which are the lean angle δ and the cut-off angle λ. The lean angle δ describes the rotation of the blade around an axis which goes through the centerpoint of the blade section at the bottom disc and which is tangential to the blade at that point. Positive values of δ refer to lean towards the suction side whereas negative values refer to lean towards the pressure side. The cut-off angle λ is applied to the leading edge of the unwound blade. The two legs of this angle are the original leading edge of the uncut blade and the cut edge which forms the new leading edge. The intersection between those two legs is always at the bottom disc, i.e. the original leading edge position only persists at the bottom disc while material is increasingly cut towards the shroud. Due to lean and cutting the leading edge, the actual blade angles β_{b1} and β_{b2} become variable over the blade height and differ from the original definition used to determine the centerpoint and the radius of the circular arc which forms the blade. Nevertheless, the blade keeps its circular shape which is important to facilitate cheap manufacturing techniques.

From the aforementioned parameters, D_2 and S are not varied by DoE but are constant or a function of other geometrical parameters, respectively. The reason for holding D_2 constant is that we consider the dimensionless aerodynamic performance which is independent of D_2 - as long as the effect of Reynolds number is small. This also applies to the rotational speed N and the fluid density ρ which are held constant, too. The reason why S is no independent parameter is that the blade thickness only has a minor effect on the aerodynamic performance and is usually selected for constructive reasons. We here assume that the blade thickness increases with increasing ratio D_1 / D_2:

$$\frac{S}{D_2} = 0.006\sqrt{\frac{D_1}{D_2}} \tag{4}$$

Further geometrical parameters are required to describe the inflow nozzle. Since the inflow nozzle is less relevant with respect to the aerodynamic performance, those parameters are a function of the impeller parameters and do not need to be varied by DoE. The inflow nozzle has a simple geometry and is fully described by its outflow diameter and the radius r_n. All dependent parameters are listed in Tab. 2.

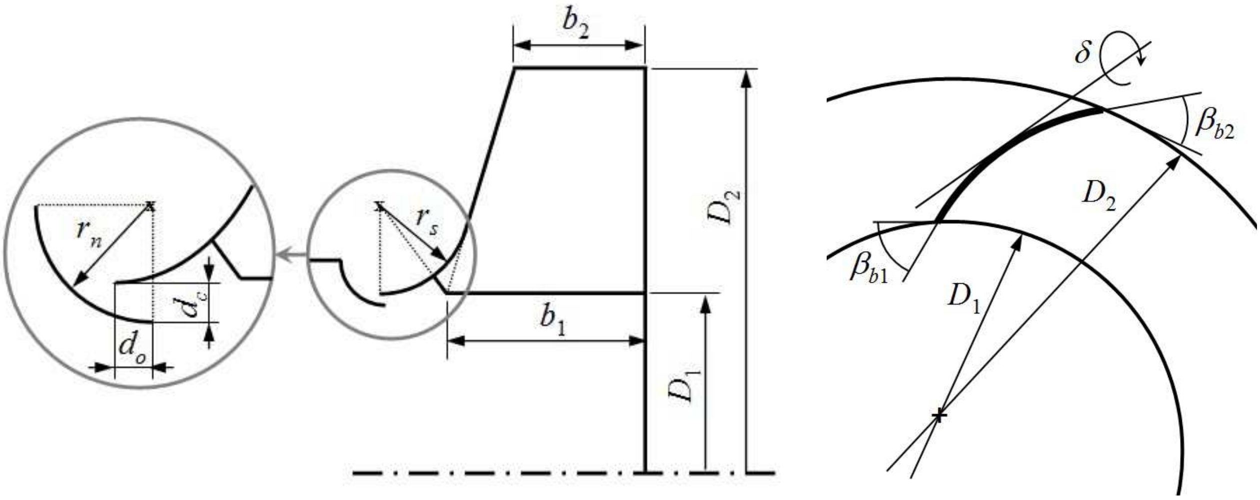

Figure 1: Side view and top view of the impeller with indication of the geometrical parameters

Table 1: independent geometrical parameters

Description	**Symbol// Definition**	**Min. Value**	**Max. Value**
Number of blades[1]	z	5	16
Diameter ratio	D_1/D_2	0.25	0.8
Inlet blade angle	β_{b1}	20°	60°
Inlet blade angle	β_{b2}	20°	60°
Inlet width ratio[2]	b_1/D_2	0.025	0.4
Outlet width ratio[2]	b_2/D_2	0.025	0.4
Shroud radius ratio	r_s/D_1	0.14	0.3
Lean angle	δ	-15°	15°
Cut-off angle	λ	0°	30°

[1] only integers are possible
[2] b_2 must always be smaller than or equal to b_1

Table 2: dependent and constant geometrical parameters

Description	**Symbol// Definition**	**Value**
Outer diameter	D_2	0.3 m
Rotational speed	N	50 s^{-1}
Fluid density	ρ	1.2 kg/m^3
Blade thickness ratio	S/D_2	see eq. (4)
Nozzle radius ratio	r_n/D_1	0.25
Clearance ratio	d_c/D_1	0.02
Overlap ratio	d_o/D_1	0.03

Impellers for validation experiments and experimental set-up

Experimental validation of the CFD model is performed with a total of seven prototypes. A first series of three prototypes (named VAL1 - VAL3) was produced with the aim of maximum geometrical diversity. The geometrical parameters of the first prototype (VAL1) are the mean values of the lower and upper limits indicated by Tab. 1 whereas VAL2 and VAL3 cover the extreme values. Those prototypes were used to validate the CFD model at an early development stage (prior to simulate thousands of impeller geometries). A second series of four prototypes (named VAL4 - VAL7) was produced after the development of the optimization scheme. VAL4 was optimized with the aim of maximum η_{ts} without constraining the design point, i.e. the optimization scheme automatically found the design point at which the achievable efficiency is maximal. After that, three other design points that differ substantially from each other and from VAL4 were selected and used as a constraint in the optimization scheme yielding the prototypes VAL5 - VAL7. Fig. 2 depicts three examples of the prototypes.

The validation experiments were performed on a chamber test rig in accordance with EN ISO 5801 [7]. The manufacturing technique used to produce the prototypes was stereolithography.

Figure 2: Prototypes VAL1, VAL2 and VAL6

CFD model and optimization of the numerical grid

The CFD model used for this work emulates the experimental set-up. The impeller sucks air from a chamber trough an inflow nozzle and exhausts into the free environment. The boundary conditions are constant mass flow rate at the chamber inlet, ambient pressure at the boundaries of the outflow area and no slip at the walls. Hub, shroud and blade are placed in a rotating subdomain, i.e. they are rotating with the rotational speed N. All other walls are stationary.

Due to the high number of CFD simulations required to train the meta-models, only stationary simulations were affordable, i.e. the Reynolds-Averaged Navier-Stokes (RANS) method was used. The solver selected was ANSYS CFX 14.5.7 and the turbulence model used was shear stress transport (SST). As usual in RANS simulations, only one blade channel was simulated and rotational periodicity was assumed at the lateral boundaries. The interface type in between the rotating blade and the stationary environment was frozen rotor.

The computational grid of the impeller was generated with ANSYS TurboGrid 14.5.7 and the grid of the environment was generated with ICEM CFD. The quality of the CFD results strongly depends on the size of the computational domain, the number of nodes and their distribution in the domain. Those parameters were optimized in an earlier study [8]. To this end, an objective function was defined which contains one term for measuring the inaccuracy and another term measuring the numerical expense. This objective function was minimized with the Simplex method [9]. The accuracy was estimated by comparing the CFD results with experimental results in terms of ψ_{ts} and η_{ts}. This comparison was performed for the prototypes VAL1, VAL2 and VAL3. The numerical cost of a simulation was assumed to be proportional to the overall number of nodes. Details about the grid optimization and the outcome can be found in [8]. One of the essential results is that the optimal number of nodes is 650,000 provided that they are distributed according to the optimization results.

Design of experiment

One crucial step in building a metamodel is the design of experiments (DoE). It determines which geometric parameter combinations should be investigated by CFD simulations. As recommended by Santner et al. [10], a space filling design should be chosen if no prior knowledge about the process is available. Due to its advantages and simplicity [10] a latin hypercube (LH) design was used. The non-collapsing property of LH designs is one advantage compared to commonly used grid designs. Non-collapsing means, that if all data points are projected onto one axis, all values along that axis only occur once. In order to achieve good space filling properties the LH design was optimized with the extended deterministic local search method described by Ebert et al. [11].

After the existence of preliminary metamodels based on the first CFD results, a second DoE called the "active learning phase" was performed. In contrast to the first DoE, the focus was on aerodynamics rather than geometrical diversity. New points were primarily added in those areas of the input space that are relevant with respect to aerodynamic optimization or in areas with poor performance of the preliminary metamodels. All details about the active learning phase can be found in [13].

Training of the metamodels

Two meta-model types were tested: Local Model Networks (LMN) and Multi-Layer Perceptrons (MLP). A general description of LMNs and MLPs is given by Nelles [14]. A more detailed descriptions how these metamodels are applied for optimization of centrifugal impellers can be found in [8]. Both types were used to predict characteristic fan curves based on a set of geometrical impeller parameters (named the input). There are basically two ways to predict characteristic fan curves. The simplest approach is a direct prediction of the quantities of interest (e.g. ψ_{ts} or η_{ts}). The disadvantage of that strategy is that these quantities are dependent on the flow coefficient φ which therefore has to be considered as an additional input to the metamodels. This strategy not only increases training time, but also increases the complexity of the metamodels which eventually increases the risk of overfitting effects. An alternative is to use parameterized shapes of the characteristic curves. In that case, the metamodels are used to predict the curve parameters and the characteristic curve is built on the basis of these curve parameters. One essential advantage of that method is that the curve parameters are only dependent on the impeller geometry. As a consequence, φ is no metamodel input. Another advantage of this method is that the curve parameters can be physically interpretable quantities such as the flow coefficient at zero ψ_{ts} or maximum η_{ts}. This allows direct control over the most important points of the characteristic curve. The main disadvantage of this method is the loss of flexibility since not all curve shapes can be emulated by the pre-defined functions in an adequate way. A further problem is the accumulation of errors if more than one curve parameter is predicted imprecisely.

Altogether, four ways to predict the aerodynamic performance were tested and compared: LMN or MLP combined with either a direct prediction of performance using φ as an input or an indirect prediction of performance via parameterized characteristic curves. Eventually, it was found that the most reliable performance prediction is obtained using the weighted mean value of all four performance predictions. The weighting factors are proportional to the inverse rout mean square error of the corresponding metamodel on test data.

Optimization algorithm

The aerodynamic optimization is performed with an evolutionary optimization algorithm. The objective function consists of the total-to-static efficiency and penalty terms that ensure the fulfilment of case-specific constraints. Exemplarily, eq. (5) represents the objective function of an optimization problem with given design point but without further operational or geometrical constraints:

$$OF = \eta_{ts} + w \cdot \left| \psi_{\text{target}} - \psi_{\text{actual}} \right| \tag{5}$$

ψ_{target} is the targeted design pressure coefficient. η_{ts} and ψ_{actual} are predicted by the metamodels.

The weighting factor w is individual for each penalty term. On the one hand, it needs to be high enough to ensure fulfilment of the constraint. On the other hand, the penalty term must not become dominant in the sense that individuals with high efficiencies but a minor violation of the constraint die out in the first generations of the evolutionary algorithm. In the present example with constrained pressure coefficient, a magnitude of $w = 5$ was found to be adequate.

The software implementation of the optimization algorithm is an in-house code written in MATLAB™. It was inspired by the work by Thévenin and Janiga [15]. The number of individuals per generation is 2000. The generation of an offspring generation is mostly conducted with the "crossover" method and only a small portion of the offspring generation is based on the "averaging" method. Moderate and random mutations are applied after the generation of the offspring. Given these settings, the algorithm converges after a couple of hundred generations and yields repeatable results that are independent of the initialization of the first generation.

RESULTS

This section discusses the capabilities of the metamodels and the whole optimization scheme. It starts with a discussion of the range of design points that were simulated in the context of the DoE. This range represents the design space in which the metamodels are applicable. Subsequently, the quality of the metamodels is checked in two ways. Firstly, CFD is compared to experimental measurements of seven prototypes. Secondly, the metamodel prediction of optimized impellers is compared to CFD results. The section concludes with a discussion of the efficiency obtained by the optimization.

Feasible design points

It is commonly agreed that not all design points can be realized with centrifugal fans. Cordier [16] found that there is a correlation between the specific fan speed σ

$$\sigma = \frac{N}{\left(2\pi^2\right)^{-\frac{1}{4}} \left(\frac{\Delta p_{tt}}{\rho}\right)^{\frac{3}{4}} Q^{-\frac{1}{2}}} \tag{6}$$

and the specific fan diameter δ

$$\delta = \frac{\Delta p_{tt}}{\left(\frac{8}{\pi^2}\right)^{\frac{1}{4}} \left(\frac{\Delta p_{tt}}{\rho}\right)^{-\frac{1}{4}} Q^{\frac{1}{2}}} \tag{7}$$

which limits the achievable design points to a narrow band in the σ-δ diagram. Moreover, not all specific fan speeds are suitable for centrifugal fans. Carolus [17] presents a σ-δ diagram in which the classic realm of centrifugal fans is assumed to be in the range $0.1 \leq \sigma \leq 0.5$. Smaller magnitudes are typically realized by positive displacement machines, larger magnitudes are typically realized by mixed-flow or axial fans.

Fig. 3 depicts a σ-δ diagram in which the typical correlation between σ and δ is illustrated by a black curve based on a formula by Pelz [18]. This curve is known as the Cordier curve. The grey area represents the range of the operating points that were simulated in the context of the DoE. It forms a band around the Cordier curve in the range $0.07 \leq \sigma \leq 2$. Hence, it is confirmed all typical design points of centrifugal fans can be realized with the present input space. In fact, the design space is even extended since much smaller or larger specific fan speeds than usually used for centrifugal fans can be realized, too.

In addition, Fig. 3 depicts the operating points with the highest total-to-static efficiency of the seven fans used for validation. As intended, the specific fan speeds of those points are quite different and range from $\sigma = 0.08$ to $\sigma = 0.8$.

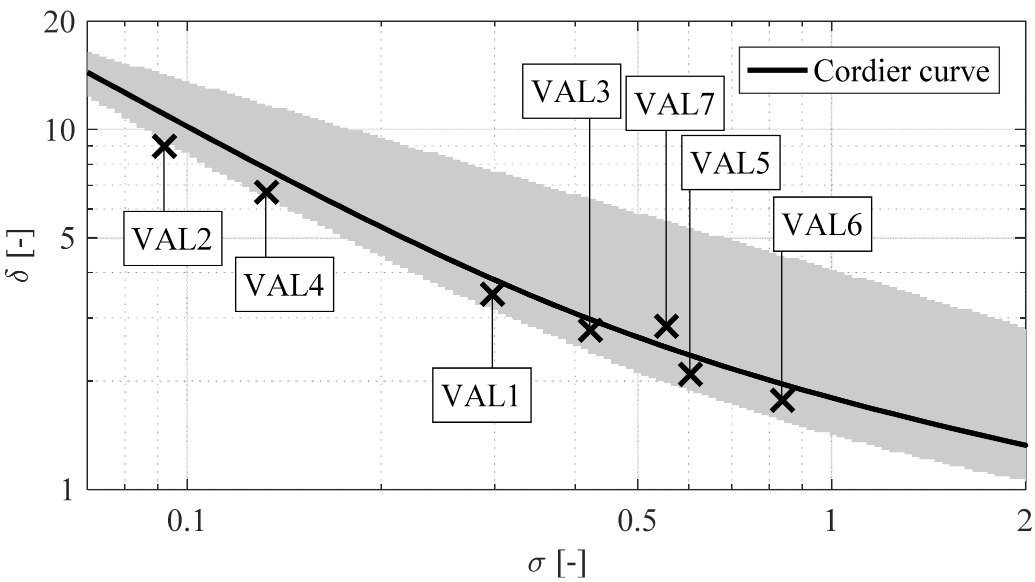

Figure 3: σ–δ diagram with indication of all operating points simulated in the context of the DoE (grey area) and the optimal points of the fans used for validation (black crosses)

Validation of the CFD model

Fig. 4 compares the CFD-simulated and measured characteristic curves of the seven validation examples in terms of ψ_{ts} and η_{ts}. Not all CFD-simulated operating points are depicted since some results are considered unreliable. This applies to all operating points with strong secondary flows and a highly instationary flow field which cannot be computed adequately with the RANS equations. Two criteria were defined to detect unreliable CFD results. The first criterion deals with the area-averaged wall shear stress τ_w on the blade which should not fall below a critical value. The second criterion deals with radial backflow in the blade channel. In an ideal flow field, there is only flow from the inside to the outside. Secondary flows, however, can lead to local backflow. More details on the sorting of unreliable operating points can be found in [8]. Note that only operating points that fulfil the two criteria were used for the training of the metamodels.

As it can be observed in Fig. 4, the agreement between CFD and experiment is very good for operating points which do not violate the two criteria for instationary flow. It is hence concluded that the

CFD model is successfully validated. In particular, it is important that the good agreement also applies to the optimized fans proving that the optimization scheme actually found an aerodynamic optimum instead of just exploiting a weak point of the CFD model.

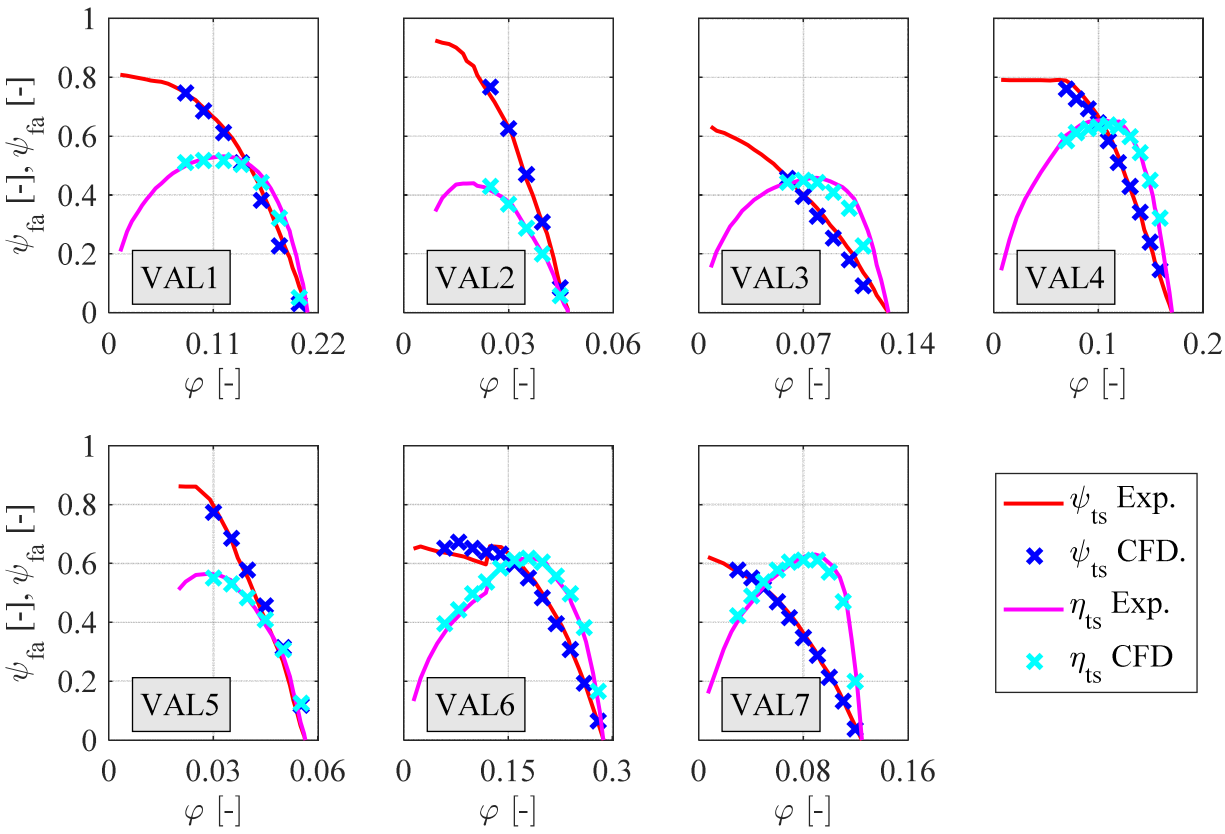

Figure 4:Comparison between CFD and experiment for the seven prototypes name VAL1-VAL7

Suitability of the metamodels for optimization

Metamodels of very high quality are required if they are to be used to evaluate the target function in optimization algorithms. The reason is that any weak point of the metamodel (where the predicted efficiency is unrealistically high) will be exploited by the optimization algorithm instead of converging to the real aerodynamic optimum. The assessment if the present metamodels are suitable for optimization was performed by conducting optimizations with the target of maximum η_{ts} for various design points (see eq. (5)). The resulting geometries were then simulated by means of CFD. Fig. 5 compares the metamodel prediction to the CFD result. Obviously, the metamodel prediction is very reliable for standard design points, i.e. design points close to the Cordier curve. An increasing uncertainty is observed at untypically small specific fan speeds ($\sigma < 0.1$) and if the specific diameter significantly higher than suggested by the Cordier curve.

Achievable efficiency

Fig. 6 depicts the CFD-predicted total-to-static efficiency of the optimized impellers in a σ-δ diagram. As expected, the highest efficiencies can be achieved near the Cordier curve and at typical specific speeds of centrifugal fans. Given the present parameterization, the most efficient impellers feature total-to-static efficiencies around 64 %. Naturally, the achievable efficiency strongly decays at design points far from the Cordier curve. Additionally, a moderate decay of achievable efficiency

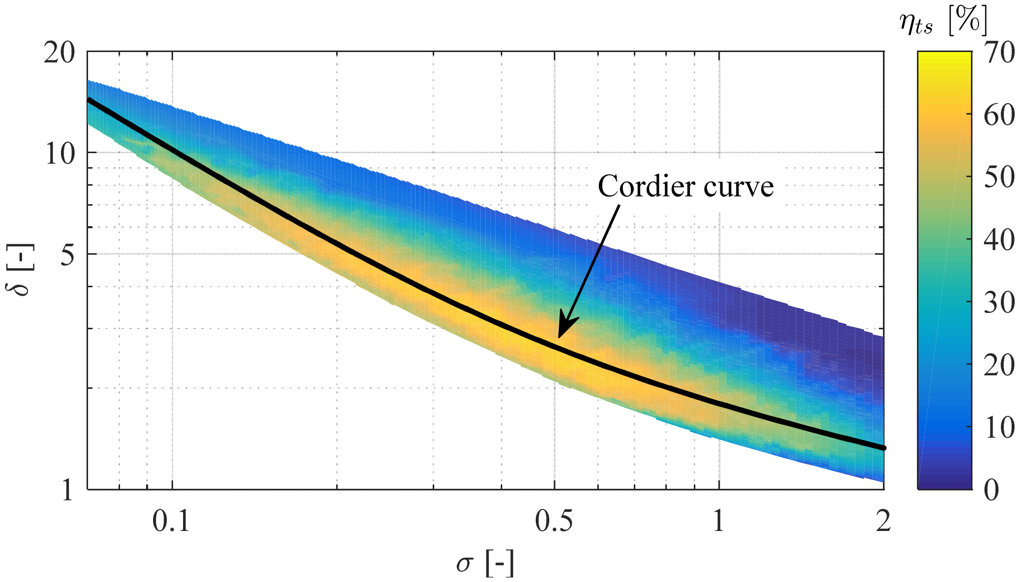

Figure 6:CFD-simulated total-to-static efficiency of optimized impellers

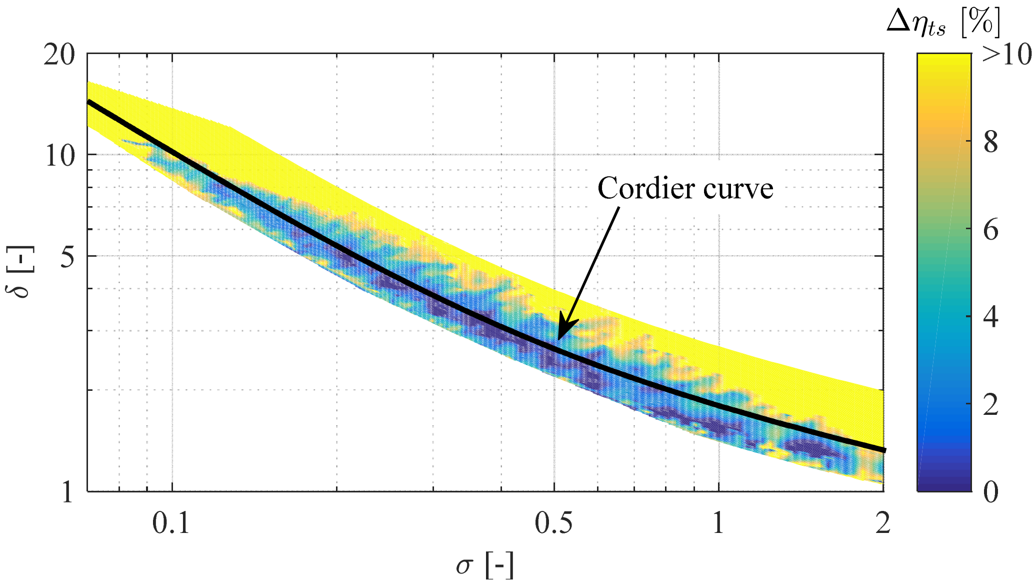

Figure 5: Difference between the total-to-static efficiency predicted by the metamodels and by CFD of optimized impellers

is observed at specific fan speeds that are more typical for mixed-flow or axial fans rather than for centrifugal fans.

Since the focus of this work is on total-to-static efficiency, the overall losses contain a contribution from the exit losses. Fig. 7 depicts the relative share of the exit losses L_{ex} in the overall losses L_{ov}. It is highest for low values of both σ and δ. Such design points are associated with relatively low circumferential tip speeds wherefore higher circumferential flow velocities are required according to Euler's equation of turbomachinery. In addition, the small values of δ lead to higher meridional velocities which also contribute to the exit losses. Towards other design points, the relevance of the exit losses strongly decays for two reasons. Firstly, the absolute value of the exit losses decreases when using larger and/or faster rotating fans. Secondly, the relative share of the exit losses is further reduced at design points with high friction losses or high volumetric losses. Internal friction plays a dominant role at design points that are far from the Cordier curve or have specific fan speeds that are more typically realized by other fan types. In addition, the external friction between the bottom disc and shroud and the surrounding air is relevant at very large specific fan diameters. The efficiency at those design points is further reduced by relatively high volumetric losses.

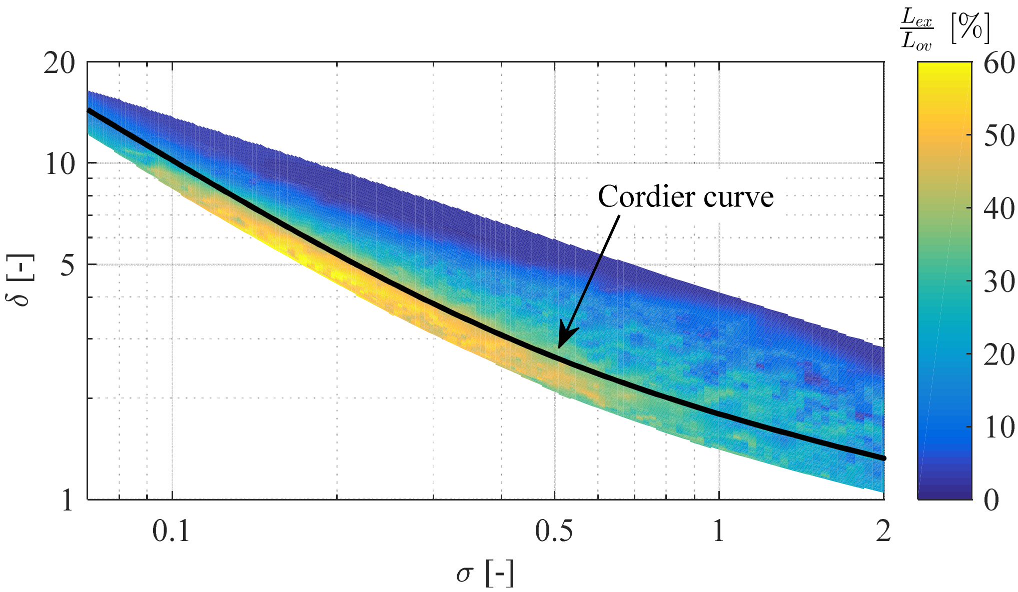

Figure 7: Relative share of the exit losses in the overall losses

CONCLUSIONS

A quick and reliable optimization method for centrifugal impellers was demonstrated. The quickness stems from replacing CFD simulations by CFD-trained metamodels which evaluate the aerodynamic objective functions many orders of magnitude faster than CFD itself. The reliability was tested by optimizing impellers for various design points and performing subsequent CFD simulations of the optimized fans. The tested design points cover the complete typical realm of centrifugal fans according to the Cordier diagram. In addition, the typical realm of mixed-flow and axial fans is partly covered, too. At most design points, CFD confirms well the metamodel prediction which proves the reliability of the method. Moreover, experimental validation was performed with seven prototypes designed with the aim of maximum geometrical and operational diversity. Due to the good agreement between CFD and experiment, the CFD model is considered successfully validated.

Current research focuses on the practical application of the new optimization method which generally involves a set of geometrical constraints such as limited axial depth, maximum number of blades, structural health, etc. While the optimization method presented in this paper is basically suitable to handle such constraints (without the development of new metamodels), the quality of the optimization results is only proven for unconstrained optimization so far. The necessity to change the geometrical parameter space is another potential challenge in practical optimization. For instance, the practical inflow conditions might differ from the conditions assumed in the CFD simulations used to train the metamodels. In such cases, the new optimization method only yields preliminary results that can be used as the basis for further modifications. This also holds true for Reynolds numbers that differ significantly from the Reynolds numbers assumed in the CFD simulations used to train the metamodels.

ACKNOWLEDGEMENTS

This work was funded by the German Ministry for Economic Affairs and Energy (BMWi) based on a decision of the Germany Bundestag (federal parliament).

REFERENCES

[1] Pfleiderer, C., **1961**, Die Kreiselpumpe für Flüssigkeiten und Gase, *Springer Verlag*, Berlin-Heidelberg.
[2] Pfleiderer, C., Petermann, H., **1991**, Strömungsmaschinen, *Springer-Verlag*, Berlin, Germany.
[3] Bommes, L., Fricke, J., Grundmann, R., **2003**, Ventilatoren, *Vulkan Verlag*, Essen.
[4] Ratter, H., Caglar, S., Gabi, M., **2012**, "A Combination of Conventional Layout Desing and Numerical Methods for the Optimization of Centrifugal Fans", *Proc. 14th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, ISROMAC-14*, Honolulu, Hawaii, USA.
[5] Ratter, H., Çağlar, Ş., Gabi, M., **2013**, "Empirical model for the quantitative prediction of losses of radial fans based on CFD calculations", *Proc. 11th International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows*, Shenzhen, China.
[6] Ratter, H., Çağlar, Ş., Gabi, M., **2014**, "A Coupled Blade Adjustment and Response Surface Method for the Optimization of Radial Fans Without Housing", *Proc. ASME TurboExpo 2014*, Düsseldorf, Germany.
[7] EN ISO 5801:2009, **2010**, "Industrial Fans - Performance Testing Using Standadized Airways," *Beuth Verlag, Berlin, Germany*, Berlin, Germany.
[8] Bamberger, K., Belz, J., Carolus, T., Nelles, O., **2016**, "Aerodynamic Optimization of Centrifugal Fans Using CFD-Trained Meta-Models", *Proc. International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC) 2016*, Honolulu, Hawaii, USA.
[9] Nelder, J. A., Mead, R., **1965**, "A Simplex Method for Function Minimization", *Computer Journal*, 7, pp. 308-313.
[10] Santner, T., Williams, B., Notz., W., **2003**, The design and analysis of computer experiments, *Springer Science & Business Media*.
[11] Ebert, T., Fischer, T., Belz, J., Heinz, T. O., Kampmann, G., Nelles, O., **2015**, "Extended Deterministic Local Search Algorithm for Maximin Latin Hypercube Designs", *Proc. IEEE Symposium on Computational Intelligence in Control and Automation*.
[12] Belz, J., Bamberger, K., Nelles, O., **2016**, "Order of Experimentation for Metamodeling Tasks", *Proc. International Joint Conference on Neural Networks (IJCNN)*, Vancouver, Canada.
[13] Hartmann, B., Ebert, T., Nelles, O., **2011**, "Model-based Design of Experiments Based on Local Model Networks for Nonlinear Processes with low Noise Levels", *Proc. American Control Conference*.
[14] Nelles, O., **2001**, Nonlinear System Identification, *Springer Verlag GmbH*, Heidelberg, Germany.
[15] Thévenin, D., Janiga, G., **2008**, Optimization and Computational Fluid Dynamics, *Springer Verlag GmbH*, Heidelberg, Germany.
[16] Cordier, O., 1953, "Ähnlichkeitsbedingungen für Strömungsmaschinen", *BWK*, 5(10), pp. 337-340.
[17] Carolus, T., **2012**, Ventilatoren - Aerodynamischer Entwurf, Schallvorhersage, Konstruktion, *Springer Vieweg*, Wiesbaden, Germany.
[18] Pelz, P., Metzler, M., **2012**, "Optimization of Power-Specific Investment Costs for Small Hydropower", *Proc. 17th international Seminar on Hydropower Plants*.