Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science

<http://pic.sagepub.com/>

Design of radial impellers: A combined extended analytical and numerical method

P Épple, B Karic, C Ilic, S Becker, F Durst and A Delgado

Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science 2009 223: 901

DOI: 10.1243/09544062JMES1196

The online version of this article can be found at:

<http://pic.sagepub.com/content/223/4/901>

Published by:

<http://www.sagepublications.com>

On behalf of:

[Institution of Mechanical Engineers](#)

Additional services and information for *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* can be found at:

Email Alerts: <http://pic.sagepub.com/cgi/alerts>

Subscriptions: <http://pic.sagepub.com/subscriptions>

Reprints: <http://www.sagepub.com/journalsReprints.nav>

Permissions: <http://www.sagepub.com/journalsPermissions.nav>

Citations: <http://pic.sagepub.com/content/223/4/901.refs.html>

>> [Version of Record](#) - Apr 1, 2009

[What is This?](#)

Design of radial impellers: a combined extended analytical and numerical method

P Eppel^{1*}, B Karic¹, Č Ilić¹, S Becker¹, F Durst², and A Delgado¹

¹Institute of Fluid Mechanics, LSTM, Friedrich-Alexander University, Erlangen-Nuremberg, Erlangen, Germany

²Centre of Advanced Fluid Mechanics, FMP Technology GMBH, Erlangen, Germany

The manuscript was received on 27 May 2008 and was accepted after revision for publication on 7 October 2008.

DOI: 10.1243/09544062JMES1196

Abstract: The use of high-speed radial impellers is very common in blowers for industrial application. It is also very common to manufacture these impellers using circular arc blades. The design process as well is almost always based on former impeller series and experimental data available. In this work, a method is presented to improve the efficiency of radial impellers with a combined analytical and numerical method. This method is based on an extended analytical formulation of the flow in radial impellers, allowing optimizing efficiency in the design stage. It is complemented by the mathematical implementation of a well-known qualitative principle of efficiency optimization according to Carnot. Finally, the torque-speed characteristic of the motor is included in the design stage. The blade shapes are computed using an inverse method. The design is then validated by means of computational fluid dynamics (CFD) computation with a commercial solver. Finally, a prototype was built and measurements were carried out in a test rig. It is also shown that the design method provided very good predictions leading to an efficiency increase of 13 per cent and a maximum flowrate increase of 11 per cent. The design point was also met. It is also shown that the numerical computations and measurements are in good agreement. An analysis of the CFD results is also presented, giving an insight view into the substantial flow information within the old and the new impellers. The method presented is a combined analytical and numerical method suited to design high-efficiency radial impellers considering also the torque-speed characteristic of the motor without the need of a previous impeller series or knowledge of experimental data.

Keywords: computational fluid dynamics, inverse design method, integrated performance optimization, mean line analysis, radial impeller, test rig measurements, torque-speed characteristic

1 INTRODUCTION

Nowadays, the development of turbomachinery is still performed in most of the cases by improving existing geometries experimentally and/or applying simple design methods and rules out of the text books [1–4]. The experience of the designer also plays a major role in this process. Increasingly, computational tools for the simulation of the flows in turbomachines are being utilized for turbomachine

development. Even then the simulation results are only as good as the geometry input for simulation. Therefore, this tool alone is not enough to achieve substantial efficiency improvements. Improvements have been achieved with the aid of neural networks, genetic algorithms combined with proper parameterization [5]. Another approach, which is based on the in-depth understanding of the flow physics inside the impeller, was developed in this work. This method is capable of providing with a few designs substantial performance improvements for high-end industrial impellers. The methods described by Van den Braembussche [5] can then still be applied to bring about a final improvement. One has to mention that these later methods require much more simulations and computational capacity than the method described in the present work.

**Corresponding author: Institute of Fluid Mechanics, LSTM, Friedrich-Alexander University, Erlangen-Nuremberg, Cauerstrasse 4, Erlangen 91058, Germany.
email: philipp.eppel@lstm.uni-erlangen.de*

The present method was developed to perform a well-directed design of centrifugal impellers and impeller–diffuser units with a limited number of designs and computational fluid dynamics (CFD) simulations, since the latter are very costly. The method is based on a proper understanding of the flow physics and reformulation of the existing formulae in the literature (e.g. Bohl and Elmendorf [1], Eck [2], Eckert and Schnell [3], and Sigloch [4]), in combination with a well-known qualitative principle formulated by Carnot, which is mathematically reformulated in this work.

This work presents the method and gives a practical impeller design example. How to perform a variable speed impeller performance optimization is shown: a design coupled with CFD verification. Prototypes were built to confirm the new design. A very good prediction of the measured values was achieved by the design method combined with CFD verification. It is shown that with the present method efficiency improvements of about 13 per cent were achieved accompanied by a substantial pressure flowrate characteristic improvement.

2 EXTENDED DESIGN METHOD

The fundamental turbomachinery equation according to Euler for incompressible flows is given by

$$\Delta p_{t,12} = \rho(u_2 c_{u2} - u_1 c_{u1}) \quad (1)$$

which gives the total pressure increase between the inlet and the outlet of the impeller as a function of the fluid density, the peripheral velocities of the impeller at the inlet and the outlet, and the tangential absolute velocity of the air at the inlet and the outlet. At radial entry, which is the case, this equation reduces to

$$\Delta p_t = \rho u_2 c_{u2} \quad (2)$$

Referring to the velocity triangle at the outlet, Fig. 1, this equation can be finally rewritten as

$$\Delta p_t = \frac{1}{2} \rho (u_2^2 - w_2^2 + c_2^2) \quad (3)$$

Fig. 1 Velocity triangle at the outlet

It is important to mention that in Fig. 1 blade congruency is assumed and hence the flow angle γ_2 is assumed to be equal to the blade angle β , i.e. $\gamma_2 = \beta_2$. Therefore, all formulae in this section are derived using the blade angle β_2 . In this fundamental analysis, any deviation of the flow angle from the blade angle (e.g. through a slip factor [6]) will not be considered. Due to the radial entry, the total pressure increase is a function of the outlet velocities only. It can be split into dynamic and static pressure increase

$$\Delta p_d = \frac{1}{2} \rho c_2^2 \quad (4)$$

$$\Delta p_s = \frac{1}{2} \rho (u_2^2 - w_2^2) \quad (5)$$

Unlike the traditional literature [1–4], these formulae do not represent the ‘gap pressures’ (from the German term ‘Spaltdruck’), as formulated by Eck [2], i.e. the pressure rise between the inlet and the outlet of the impeller

$$\Delta p_{d,12} = \frac{1}{2} \rho (c_2^2 - c_1^2) \quad (6)$$

$$\Delta p_{s,12} = \frac{1}{2} \rho [(u_2^2 - u_1^2) + (w_1^2 - w_2^2)] \quad (7)$$

but the pressures as measured in a test rig. Considering that

$$u_2 = \pi d_2 n \quad (8)$$

and

$$w_2 = \frac{c_{m2}}{\sin \beta_2} = \frac{Q}{\pi d_2 b_2 \sin \beta_2} \quad (9)$$

with

$$c_{m2} = \frac{Q}{\pi d_2 b_2} \quad (10)$$

the static pressure rise, equation (5), can be written as

$$\Delta p_s = \frac{1}{2} \rho \left[(\pi d_2 n)^2 - \left(\frac{Q}{\pi d_2 b_2 \sin \beta_2} \right)^2 \right] \quad (11)$$

The maximum static pressure at zero flowrate is given by

$$\Delta p_{s,\max} = \frac{1}{2} \rho u_2^2 = \frac{1}{2} \rho (\pi d_2 n)^2 \quad (12)$$

and the maximum flowrate at zero static pressure by

$$Q_{\max} = \pi^2 d_2^2 b_2 n \sin \beta_2 \quad (13)$$

These equations allow, at the design stage, to set the proper characteristic of pressure against flowrate. From equation (12), the maximum static pressure is

Fig. 2 Static head coefficient against flow coefficient

proportional to the square of the outlet diameter d_2 and the square of the speed n . The maximum flowrate is proportional to the second power of the outlet diameter d_2 and directly proportional to the outlet height b_2 , the speed n , and the sinus of the outlet angle β_2 , in equation (13). Equation (5) can be rewritten as a static head coefficient as shown in Fig. 2

$$\psi_s \equiv \frac{\Delta p_s}{\rho u_2^2} = \frac{1}{2} \left[1 - \left(\frac{\varphi}{\sin \beta_2} \right)^2 \right] \quad (14)$$

where

$$\varphi = \frac{c_{m2}}{u_2} \quad (15)$$

is the flow coefficient.

The maximum static head and flowrate is given for impellers with an outlet angle of $\pi/2$. Setting equation (14) to zero, one obtains the maximum flow coefficient

$$\varphi_{\max} = \sin \beta_2 \quad (16)$$

The total head coefficient is given by

$$\psi_t = \frac{\Delta p_t}{\rho u_2^2} = \frac{\rho u_2 c_{u2}}{\rho u_2^2} = \frac{c_{u2}}{u_2} \quad (17)$$

since

$$c_{u2} = u_2 - \frac{c_{m2}}{\tan \beta_2} \quad (18)$$

$$\psi_t = 1 - \frac{\varphi}{\tan \beta_2} \quad (19)$$

In many industrial fan applications the dynamic pressure at the outlet is not recovered, it is simply lost.

For these cases, the useful hydraulic power is given by

$$P_{\text{hyd}} = Q \Delta p_s = \frac{1}{2} \rho (u_2^2 - w_2^2) Q \quad (20)$$

Ideally, without losses, the total hydraulic power is equal to the shaft power and given by

$$P_{\text{shaft}} = Q \Delta p_t = \rho Q u_2 c_{u2} \quad (21)$$

Hence, in the ideal case, where losses are neglected, one can define the total-to-static efficiency as

$$\eta_{t-s} = \frac{P_{\text{hyd}}}{P_{\text{shaft}}} = \frac{Q \Delta p_s}{Q \Delta p_t} = \frac{1}{2} \frac{u_2^2 - w_2^2}{c_{u2} u_2} \quad (22)$$

since

$$w_2 = \frac{c_{m2}}{\sin \beta_2} \quad (23)$$

or

$$\frac{w_2/u_2}{\varphi} = \frac{1}{\sin \beta_2} \quad (24)$$

and using also equation (18) one can rewrite this equation as

$$\eta_{t-s} = \frac{1}{2} \frac{u_2^2 - (c_{m2}/\sin \beta_2)^2}{u_2 (u_2 - c_{m2}/\tan \beta_2)} \quad (25)$$

which can be rearranged in terms of the flow coefficient $\varphi = c_{m2}/u_2$ as

$$\eta_{t-s} = \frac{1}{2} \frac{[1 - (\varphi/\sin \beta_2)^2]}{(1 - \varphi/\tan \beta_2)} \quad (26)$$

The total-to-static efficiency is a function of the outlet angle and the flow coefficient only. It is also worthwhile mentioning that

$$\lim_{\varphi \rightarrow 0} \eta_{t-s} = \frac{1}{2} \quad (27)$$

One has to bear in mind that this is the limit towards zero flow coefficient of the ideal total-to-static efficiency. In a real flow with viscosity and losses this limit is zero. Equation (26) is plotted in Fig. 3.

Considering equations (8), (10), (13), and (15) one can also write equation (26) as

$$\eta_{t-s} = \frac{1}{2} \frac{1 - X^2}{1 - X \cos \beta_2} \quad (28)$$

with

$$X \equiv \frac{c_m}{c_{m,\max}} = \frac{Q}{Q_{\max}} \quad (29)$$

Equation (28) shows that the total-to-static efficiency is a function of the flowrate and the outlet

Fig. 3 Total-to-static efficiency against flow coefficient

angle β_2 only. To find the flow coefficient of maximum efficiency equation (26) has to be differentiated with respect to φ and set to zero. Hence

$$\varphi_{\max} = \begin{cases} \tan \beta_2 (1 - \sin \beta_2) & (0^\circ \leq \beta_2 \leq 90^\circ) \\ 0 & (90^\circ \leq \beta_2 \leq 180^\circ) \end{cases} \quad (30)$$

It has to be mentioned that for outlet angles $\beta_2 > 90^\circ$ the maximum of equation (26) is located at negative flowrates and hence the physically meaningful maximum lies at $\varphi = 0$.

Substituting equation (30) into equation (26) the maximum total-to-static efficiency is found to be

$$\eta_{t-s,\max} = \begin{cases} \frac{1}{1 + \sin \beta_2} & (0^\circ \leq \beta_2 \leq 90^\circ) \\ 0.5 & (90^\circ \leq \beta_2 \leq 180^\circ) \end{cases} \quad (31)$$

This equation gives the theoretical maximum of the total-to-static efficiency. It is a remarkable result that the maximum total-to-static efficiency is a function of the outlet angle β_2 only. According to this equation the maximum efficiency that can be achieved is 1, but only when the outlet angle $\beta_2 = 0^\circ$. This is of course a theoretical limit only, i.e.

$$\lim_{\beta_2 \rightarrow 0} \eta_{t-s,\max} = 1 \quad (32)$$

This limit is physically not realizable, basically for two reasons:

- (a) a blade with zero outlet angle, $\beta_2 = 0^\circ$, would simply close the impeller;
- (b) leads to infinite relative velocities w_2 (Fig. 4) and hence infinite losses due to friction.

The maximum total-to-static efficiency is plotted in Fig. 5, and the non-dimensional relative velocity, equation (24), is shown in Fig. 4. These two graphs

Fig. 4 Non-dimensional relative velocity against the outlet angle

Fig. 5 Maximum ideal total-to-static efficiency

together are central design criteria. To increase the total-to-static efficiency, it is necessary to decrease the outlet angle β_2 . In doing so, although, the relative velocity increases.

As can be seen in Fig. 4, at small outlet angles the relative velocity increases substantially. Increasing the relative velocity will also increase the losses due to friction and flow detachment. Hence, the theoretical increase in total-to-static efficiency will be cancelled out by the increased losses due to high relative velocities. It is the experience of the authors that outlet angles smaller than $\sim 15^\circ$ will not increase the total-to-static efficiency anymore.

At this design stage one has to watch equation (16) also. Reducing the outlet angle β_2 will also reduce the maximum flowrate. This can be seen also in Fig. 3.

Hence, increasing the maximum total-to-static efficiency by decreasing the outlet angle β_2 will decrease the maximum flowrate and increase the relative velocity. The proper setting of the outlet angle can be preadjusted in design stage having in mind the previous considerations and verified with a commercial CFD solver. Eventually, some iteration between design and CFD verification is needed to perform the fine-tuning of the outlet angle.

3 TORQUE-SPEED CHARACTERISTIC

The torque at the impeller shaft can be computed from the shaft power given by equation (21)

$$P_{\text{shaft}} = M_{\text{impeller}} \omega = \rho Q u_2 c_{u2} \quad (33)$$

where M is the shaft torque and $\omega = 2\pi n$ is the angular velocity. Hence

$$M_{\text{impeller}} = \frac{\rho Q u_2 c_{u2}}{\omega} \quad (34)$$

Substituting equations (8), (10), and (18) into the above equation

$$M_{\text{impeller}} = \frac{1}{2} \rho Q d_2 \left(\pi d_2 n - \frac{Q}{\pi d_2 b_2 \tan \beta_2} \right) \quad (35)$$

which is the design expression giving the explicit dependence of the torque on the geometry, given by the outlet diameter d_2 , the outlet height b_2 , and the outlet angle β_2 , the flowrate Q , and the speed n . Also, here the blade congruency of the flow is assumed, i.e. the flow angle is assumed to be equal to the blade angle ($\gamma_2 = \beta_2$). Therefore, in all equations in this section the blade angle β_2 is used. With this equation the impeller can be dimensioned to provide the proper torque at a given speed and flowrate.

The most common design case is the one with constant speed. In that case one assigns the corresponding value to the speed n and in such a way the speed does not matter anymore in further design procedure, it is given and it is constant.

However, in many cases the speed is not constant, since it is governed by the torque-speed characteristic of the driving motor. In such a case, it is necessary to consider this characteristic already in design stage.

For instance, it is very common to have universal motors as drives. Universal motors have a torque-speed characteristic of the kind

$$n = \frac{k_1}{\sqrt{M_{\text{drive}}}} + k_2 \quad (36)$$

where k_1 and k_2 are motor constants and M_{drive} is the torque of the driving motor [7]. This formula can be

solved for the torque as

$$M_{\text{drive}} = \left(\frac{k_1}{n - k_2} \right)^2 \quad (37)$$

For a general drive and hence torque-speed characteristic one can write

$$M_{\text{drive}} = f(n) \quad (38)$$

Since the torque of the motor drive has to be the same as the torque of the impeller, i.e.

$$M_{\text{drive}} = M_{\text{impeller}} \quad (39)$$

one can set equation (38) equal to equation (35)

$$f(n) = \frac{1}{2} \rho Q d_2 \left(\pi d_2 n - \frac{Q}{\pi d_2 b_2 \tan \beta_2} \right) \quad (40)$$

Assuming that $f(n)$ is given (i.e. the torque-speed characteristic of the motor drive is known), this is a unique equation for the speed n as a function of the impeller geometry and the flowrate.

In many cases, regardless of the precise torque-speed-characteristic function, the torque-speed characteristic can be approximated satisfactorily by a straight line in the range of interest. Assuming

$$f(n) = A_{\text{drive}} + B_{\text{drive}} n \quad (41)$$

and substituting into equation (40) and solving for n

$$n = \frac{2A_{\text{drive}} + (\rho Q^2 / \pi b_2 \tan \beta_2)}{(-2B_{\text{drive}} + \pi \rho d_2^2 Q)} \quad (42)$$

With this equation, once the motor torque-speed characteristic is given, one can predict the speed for a given impeller geometry and flowrate. This speed has then to be introduced into the equations, e.g. of static pressure (11), the total-to-static efficiency (22), the torque (35), and so on to provide the correct values at the design stage.

More complicated torque-speed characteristics like the one of universal motors, equation (37), or any other characteristic can always be readily solved numerically. In most cases, however, equation (42) will be precise enough to perform a variable speed design.

4 HISTORICAL REMINDER: THE CARNOT PRINCIPLES

Comte Lazare Nicolas Marguerite Carnot (May 13, 1753–August 2, 1823), the French politician, engineer, and mathematician, formulated in his main work '*Essai sur les machines en général*' (published in 1786) the following basic principles in order to achieve maximum efficiency in hydraulic machinery [8].

1. The fluid has to transfer as much energy as possible onto the machine. It should remain only as much flow energy as necessary to allow the fluid to flow out of the machine.
2. The transfer of potential energy (static pressure) and kinetic energy (dynamic pressure) should occur in such a way that there is no shock on machine parts.
3. The fluid flow should be such that there is no decay, no turbulence, and no sudden velocity reduction.
4. In a system any movement should not be created that does not serve the final purpose.

As an example, he cites the filling of an elevated tank with a pump. In order for the fluid to get into the tank it does not need any substantial velocity.

These principles were formulated for hydraulic machinery (i.e. for power subtracting flow machines [9]), but they can also be applied to power adding machines, such as fans. One can reformulate these principles as follows:

- (a) principles 1 and 4: in a turbomachine, there should be allowed only as much velocity as it is absolutely needed to achieve the necessary pressure and flowrate;
- (b) principle 2: the shock losses [10] in the machine should be kept as low as possible;
- (c) principle 3: flow detachments, turbulence, and sudden expansions should be avoided.

In design practice, principles 1 and 4 can be translated mathematically in such a way that the relative velocity in the impeller is kept as low as possible to achieve the needed pressure and flowrate.

To consider principles 2 and 3 in design stage, the blade shape is designed by an inverse method prescribing smooth velocity and pressure curves from the impeller inlet to the impeller outlet.

The authors are well aware that these principles are ≈ 200 years old and have been incorporated into turbomachinery design rules and procedures a long time ago. They are mentioned here just as a historical reminder and for the purpose of demonstrating that, even with simple design formulae, it is possible to include these principles in a quantitative way (Carnot stated them only qualitatively). Readers interested in a modern review on these principles are kindly requested to refer to the publication of Casey [11], especially to section 4, 'CFD aerodynamic design rules', which is of major importance in this context. Of course, the publication of Casey [11] is not limited to the Carnot principles. As an example of quantitative application of these principles, the discussion regarding Figs 2 to 4 should be mentioned. Furthermore, the 'example of design' given in section 6 shows that even today there are industrial designs that do not fully implement these principles. This is not an exception, for presently it is still common to have industrial

centrifugal impellers with one circular arc and two circular arc blades: both of them do not comply with the Carnot principles, as is shown in section 5.

5 BLADE SHAPE COMPUTATION

In the previous sections, the theory of performance optimization for radial fans was presented. This theory is based on the Euler pump and turbine equation, but was extended in a way to allow performance optimization at design stage, since a total-to-static efficiency formula was derived (equation (26)). For the theory based on the Euler pump and turbine equation, it is a remarkable fact that only the inlet and outlet conditions of the impeller are relevant, since this theory is based on a balance of angular momentum between the impeller inlet and the impeller outlet. This means that for the inviscid Euler theory the blade shape is not at all relevant, what matters are only the main dimensions at the inlet (i.e. d_1 , b_1 , and β_1) and the main dimensions at the outlet, i.e. d_2 , b_2 , and β_2 . Furthermore, as was shown in section 2, because of the radial entry, only the main dimensions at the outlet (i.e. d_2 , b_2 , and β_2) are of relevance.

The basic function of a blade is to guide the flow. That means to impose to the flow a certain flow direction. Mathematically, this means that the blade angle at a certain radius $\beta(r)$ should be imposed to the flow, guiding the flow to follow the same direction, i.e. that the flow angle $\gamma(r)$ is equal to the blade angle

$$\gamma(r) = \beta(r) \quad (43)$$

The closer the blade can impose the blade angle to the flow, the better the blade. In other words, the good blade will impose the blade angle to the flow as close as possible, with losses kept to a minimum.

Industrial blades are still often circular arc blades. These blades have the advantage to be easy to compute and to manufacture. For circular arc blades, once the inlet and outlet radius and angles are defined, there is only one blade curvature radius and one wrap angle possible [3].

$$R = \frac{r_2^2 - r_1^2}{2(r_2 \cos \beta_2 - r_1 \cos \beta_1)} \quad (44)$$

$$x_M = -R \sin(\beta_2) \quad (45)$$

$$y_M = r_2 - R \cos(\beta_2) \quad (46)$$

The disadvantage is that there is no way to influence the pressure gradient in the blade channel; it is defined by the circular arc shape.

To achieve higher efficiencies, it is necessary to have control over the pressure gradient through the blade shape. This can be achieved by computing the blade shape inversely prescribing a proper pressure

increase in the blade passage. To specify these pressure curves, it is first necessary to define the inlet and outlet parameters as described in section 2.

The impeller blade shape is computed from a given velocity distribution, as it is described in the literature (e.g. Pantell [12] and Sigloch [4]), according to the formula

$$\sin \beta(r) = \frac{zs}{2\pi r} + \frac{Q}{2\pi r b(r)} \frac{1}{w(r)} \quad (47)$$

where z is the number of blades of the impeller and s the thickness of the blade. Solving for the relative velocity this formula can be rewritten as

$$w(r) = \frac{w_m(r)}{\sin \beta(r)} = \frac{Q}{(2\pi r \sin \beta(r) - zs)b(r)} \quad (48)$$

These equations follow directly from the velocity triangle, Fig. 1, and include the effect of the blade thickness s . Since for design the flow may be treated as incompressible, it is immediately possible to define a pressure distribution in order to get the blade angle β . In practice, however, it is more convenient to work directly with the blade angle $\beta(r)$ in equation (48), and compute and check the velocity and the pressure. The reason is to get direct control over the blade angle and the relative velocity. Particularly for the purpose of writing a computer program for blade design, it is favourable to start with the definition of a proper blade angle function. A convenient form of the blade angle is

$$\beta(r) = ar^2 + br + c \quad (49)$$

since $\beta(r_1) = \beta_1$ and $\beta(r_2) = \beta_2$ are given by the first part of the method described in section 2. Hence, in order to find all the unknowns in equation (49) only one intermediate angle is needed, i.e. $\beta(r_m) = \beta_m$. This intermediate angle is used to design the blade shape. It determines the relative velocity, the static and the total pressure increase between the inlet and the outlet, as well as the blade wrap angle. It is a crucial design parameter.

The hub is assumed to be flat and for the shroud shape $b(r)$ a function is assumed to result in a proper meridional area and hence a proper meridional velocity distribution. The basic definition equation for the blade height is the equation for the meridian velocity (10) written for an arbitrary radius r and solved for the blade height $b(r)$

$$b(r) = \frac{Q}{2\pi r c_m(r)} \quad (50)$$

i.e. by prescribing the meridian velocity the blade height $b(r)$ can be found. However, what is a proper meridian velocity? In the literature there are many suggestions. Pantell [12] and Eckert and Schnell [3]

suggest $c_m(r) = \text{const.}$, which leads to $b(r) = \text{const.}/r$. Eck [2] relates the blade height with the blade angle such that

$$\frac{w_2}{w_1} = \frac{d_1 b_1 \sin \beta_1}{d_2 b_2 \sin \beta_2} \geq \frac{d_1}{d_2} \quad (51)$$

which means

$$b_1 \sin \beta_1 \geq b_2 \sin \beta_2 \quad (52)$$

Observing this rule, the deceleration of w_1 to w_2 is kept in certain limits. Hönnmann [13] and Sentek and Fortuna [14] give detailed information on how to dimension the blade height based on measurement results. These results, however, cannot be generalized since they are valid only for particular impellers investigated experimentally.

In the present work, a different approach was developed. The shape of the shroud, which is identical to the blade height, should be placed on a streamline. To find the adequate streamline, five models were investigated: (1) two-dimensional potential motion against a plate; (2) axisymmetric potential motion against a plate (jet stagnation point); (3) viscous flow near a plane stagnation point (Hiemenz flow); (4) viscous flow near an axisymmetric stagnation point; and (5) viscous flow near an infinite rotating disc. For the first two flows there are analytical solutions in the literature, e.g. Prandtl and Tietjens [15]. The streamlines are given by $y = \text{const.}/x$ for the potential flow against flat plate and $y = \text{const.}/r^2$ for the axisymmetric potential motion against a plate. For the viscous flows (3), (4), and (5) exact similarity solutions exist, but the resulting ordinary differential equations have to be solved numerically [16]. This was done in the present work by a fourth-order Runge–Kutta method. The result is shown in Fig. 6.

Fig. 6 Streamlines of potential and viscous two-dimensional and axisymmetric flow

Due to the boundary layer and hence the displacement thickness the viscous solutions near the wall (i.e. for $x > 0.1$ in Fig. 6) are always above the corresponding potential flow solutions. As a general formula, however, all the flows can be approximately described by a function of the type

$$b(r) = \frac{A}{r^B} + C$$

(53)

where for two-dimensional viscous laminar flow, $B = 0.62$, for two-dimensional potential flow, $B = 1$, for axisymmetric viscous laminar flow, $B = 1.23$, and for axisymmetric potential flow, $B = 2$. In practice, values of B between 0.6 and 1.2 are recommended. The values of A and C are found by imposing the entry and outlet heights. In such a way the shroud (i.e. $b(r)$) is fully defined.

With the blade angle $\beta(r)$ and the shroud shape $b(r)$ defined, the relative velocity, equation (48), can be computed and analysed. Typical relative velocity curves are presented in Fig. 7: (a) circular arc blade; (b) two circular arc blades; (c) blade with linear w development; and (d) and (e) still good relative velocity development. Circular arc blades are not favourable since they decelerate and later accelerate the flow in the blade passage. Two circular arc blades will decelerate and accelerate twice the flow having a discontinuity in the acceleration and hence are not recommended. The blades with either straight (c) or slightly upwards (e) or downwards (d) curved relative velocity curves generally provide the best results, with no unnecessary flow deceleration and no abrupt velocity change.

Having chosen an adequate relative velocity and hence pressure development in the blade channel, the blade shape is computed according to the formula ([12], Fig. 8)

$$\theta(r) = \int_{r_1}^r \frac{1}{r \tan \beta(r)} dr$$

(54)

Equation (54) can be solved numerically leading to single curved blades, computed point by point.

Fig. 7 Relative velocity for different blade shapes against the radius

Fig. 8 Blade shape and blade angle β of radial impellers

6 EXAMPLE OF DESIGN

To demonstrate the method, a practical example will be given in this section. It will be shown how the performance of a traditional industrial radial impeller with circular arc blades operating on a motor drive with given torque-speed characteristic can be substantially improved by the combined analytical and numerical method presented in the previous section. A prototype was build and the results of validation by test rig measurements will be presented.

In Table 1, the main dimensions of the old design are presented in the left column. It refers to a circular arc blade impeller. In the right column, the main dimensions of the new design are shown. These dimensions will be derived below using the present method.

An additional starting information required for any design analysis or any new design is the torque-speed characteristic of the motor drive where the impellers operate. In the present case, the torque-speed characteristic of the motor was given and is shown in Fig. 9. One can see that the measured torque-speed-characteristic is approximately a straight line and hence the approximation with a linear regression is reasonable. In this way one can directly apply equation (42).

The first step when improving an existing impeller is to analyse its design. Using equations (44) to (46), and from Fig. 8 considering that

$$\beta(r) = \arctan \left(\frac{\Delta r}{r \Delta \theta} \right)$$

(55)

Table 1 Main dimensions of the old and the new designs

	Old design	New design
Inlet radius, r_1 (mm)	30	40
Outlet radius, r_2 (mm)	90	90
Inlet height, b_1 (mm)	3.5	10
Outlet height, b_2 (mm)	3.5	5.8
Inlet angle, β_1 (°)	46.9	27.0
Outlet angle, β_2 (°)	24.5	15.0
Wrap angle, θ (°)	73.0	120.0
Number of blades, z	5	6
Blade thickness, s (mm)	1	1

Fig. 9 Torque-speed characteristic

one can numerically compute the blade angle as a function of the radius. Since the reference impeller has parallel hub and shroud, the impeller height $b(r)$ is constant and therefore it is straight forward to compute the relative velocity through equation (48) and the meridional velocity using equation (10). These curves are shown in the left panel of Fig. 10. Comparing relative velocity w of the reference impeller on the left panel of Fig. 10 with curve (a) of Fig. 7, one can see that they indeed are similar, since both refer to circular arc blades. It is a remarkable fact that the relative and the meridional velocities are independent of the rotational speed and hence of the torque-speed characteristic.

This is understandable because they are computed in the relative frame-of-reference and hence in a stationary frame-of-reference as referred to the impeller. Already, here a preliminary design analysis has to be done. From the left panel of Fig. 10, the old design, one can see that the relative and also the meridional velocities are very high, at least higher than that at the outlet. Very high velocity means that the relative velocity is about 120 m/s which corresponds to a Mach number of about $M = 0.3$, since the fluid considered is air at normal conditions. Although this still can be considered incompressible, the aim of the new design should be, according to Carnots principle, to keep the relative velocity as low as possible in order to still achieve the necessary flowrate and pressure.

The design task consists of keeping the outer diameter d_2 constant, increasing the total-to-static efficiency and the maximum flowrate as well as the static pressure in the operation range between 40 and 60 l/s. This task can be fulfilled with the following design steps.

1. Increasing the total-to-static efficiency by decreasing the outlet angle β_2 according to equations (26) and (31).
2. Since the maximum flowrate according to equation (13) should be increased, the smaller outlet angle β_2 has to be compensated by a higher outlet height b_2 .
3. To decrease the relative velocity at the inlet, the inlet radius r_1 and height b_1 are increased accordingly.
4. The inlet angle β_1 was chosen in order to have shock less entry, i.e. $\beta_1 = \gamma_1 = \arctan(c_{m1}/u_1)$ at the design point of 55 l/s.

Fig. 10 Analytical relative and meridional velocity at the design point of 55 l/s as well as blade angle of the old (left) and new (right) impellers

The chosen values are presented in the right column of Table 1. These values have to be used to compute the analytical total-to-static efficiency and static pressure characteristics and to compare them with the analytical characteristics of the reference impeller to make sure that they meet the design requirements.

As can be seen in Fig. 11, the design requirement for the total-to-static efficiency is qualitatively fulfilled for the analytical computation. Nevertheless, only CFD computations will take account for losses and hence provide more reliable data.

The static pressure for the old and the new designs is almost the same, as in Fig. 12. At first view, it would appear that the new design will not provide substantially higher pressure and flowrates. One has to bear in mind, however, that there is a second factor influencing the final result except the analytical values: the losses. For the losses, precautions have been taken to

Fig. 11 Total-to-static efficiency for the old and the new designs: analytical, CFD, and measurement

Fig. 12 Static pressure for the old and the new designs: analytical, CFD, and measurement

Fig. 13 Analytical and measured speed characteristic for the old and the new designs

substantially reduce the relative velocity, as plotted in the right panel of Fig. 10. Here one can see, for instance, that at the impeller inlet the analytical relative velocity was reduced from ~ 120 to 50 m/s, which is less than half of the velocity in the old design. Considering that the friction losses are proportional to the square of the relative velocity, and also that at high velocities detachments and recirculation zones are much more pronounced, it can indeed be expected that the pressure and the flowrate will also be higher in the new design.

Looking to the analytical pressure characteristics of Fig. 12, it is clear that they are not parabolas as given by equation (11) if one considers the speed n to be constant. However, still these curves are given by this very equation, only that the speed is variable. To analytically predict the speed, one has to combine the torque-speed characteristic as given by the regression indicated in Fig. 9 with the torque-speed characteristic of the impeller, as given by equation (35), resulting in equation (42) where the coefficients A_{drive} and B_{drive} are given by the motor torque-speed characteristic, in Fig. 9, and the other constants by the old and the new designs, in Table 1. The result of this computation can be seen in the analytical curves of Fig. 13. The analytically predicted speed characteristics for both designs are almost the same. Even so they are important to predict the correct total-to-static efficiency, Fig. 11, and the static pressure, Fig. 12, for each design.

7 CAD MODEL

To have a correct numerical simulation (CFD) of the old and the new impellers, it is necessary first to define the flow domain. The flow domain consisted of the full impeller with all blade passages, the part of the test rig until the pressure tabs, and the outlet domain (Fig. 14).

Fig. 14 Flow domains: (1) test rig section, (2) impeller, and (3) outlet domain

The reason for including the test rig and the outlet domain in the numerical simulation is that in this way one can directly compare the results obtained from the numerical simulation with the measurements from the test rig. The CAD models were built with ProENGINEER.

The grids were generated using ANSYS ICEM 11.0. They consist of tetrahedral and prism elements at the walls. The grid for the old design was modelled with 2 607 576 elements, and for the new design, since it is thicker and hence has a larger volume, with 3 573 399 elements. For the old design 147 460 prism elements and for the new design 278 563 prism elements were used. These grid sizes resulted from a grid study conducted in order to assure grid size independency (Fig. 15).

To carry out the numerical simulations, a commercial finite volume-based Navier–Stokes solver was used, ANSYS CFX 11.0.

The inlet boundary condition as shown in Fig. 16 is mass flowrate and the exit boundary an opening boundary condition at ambient pressure. The specified rotational speeds for the impellers in domain 2 are the analytical values given in Fig. 13. Domains 1 and 3 are stationary. Hence, a multiple-frame-of-reference numerical calculation is performed. The interface between the different frames of reference is taken to be a frozen rotor general grid interface (GGI). GGI connections refer to the class of grid connections where the grid on either side of the two connected surfaces does not match. In general, GGI connections permit non-matching of node location, element type, surface extent, surface shape, and even non-matching of the flow physics across the connection. ‘Frozen rotor’ refers to the reference and/or pitch change, i.e. the frame of reference and/or pitch is changed but the relative orientation of the components across the interface is fixed. The two frames of reference connect in such a way that they each have a fixed relative

Fig. 15 Tetrahedral grids with prism layers for the old and new designs

Fig. 16 Boundary conditions and flow domains

position throughout the calculation. The frozen rotor model has the advantages of being robust, using less computer resources than other frame change models, and being well suited for high blade counts. The disadvantage of this model is that the transient effects at the frame change interface are not modelled. In the present case this is irrelevant, because steady flow simulations were performed.

The reason to have the full impeller resolved and not only one blade passage is that in fact, even for steady flow simulations, there is not one blade passage with a flow pattern equal to any other blade passage, as can be seen in Fig. 17. This has many consequences, especially when analysing local flow quantities, as for instance the blockage, as defined in the next section by equation (59). The differences in mass flowrate between different blade passages can be up to 5 per cent. This information is lost when performing the simulation with one blade passage only and then the blockage information is not reliable.

The adopted turbulence model is the shear stress transport (SST) model of Menter [17]. The model works by solving a turbulence–frequency-based model ($k-\omega$) at the wall and $k-\varepsilon$ in the bulk flow. A blending function ensures a smooth transition between the two models. The SST model performance has been studied in a large number of cases. In a NASA technical memorandum [18], SST was rated as the most accurate model for aerodynamic applications.

To be sure of the convergence of the solution, besides the mass, momentum, energy, turbulence kinetic energy, and turbulence frequency residuals,

monitor points were set on important quantities and locations. Monitor points were set at each boundary condition and interface to monitor the mass flowrate and pressure. Also, the torque of the impeller was monitored. Thus it was possible to assure that all flow domains were providing sufficiently stable converged mass flowrates and pressures, and that the impeller torque also converged to a stable value. This is also an indication that the boundary conditions set are physically consistent.

8 CFD RESULTS ANALYSIS

To validate the design and confirm the improvements, CFD computations were carried out. Typically, CFD computations are used to validate and to compare design parameter studies, in order to pick out the best design for prototyping. In the present case, only one iteration or design was needed, since the new design already met the design requirements. Nevertheless, in general, one does parameter variations in order to sweep a proper parameter range and to find the final geometry. With the present design method, however, in general only a few iterations are needed, because the analytical design, when performed correctly, already yield very good results for industrial applications. Yet, the CFD validation is strongly recommended and regarded as an integral part of the method.

The main results of the numerical simulation were already included in Figs 11 and 12. From Fig. 12 one can see that, although the two designs have approximately the same analytical pressure flowrate characteristics, numerical results show that the new design has much higher pressure and flowrate values. This means that the new design has much less losses and hence performs closer to the analytical model. This also reflects in the efficiency, as can also be seen in Fig. 11. The new design has a higher analytical total-to-static efficiency than the old design, which is confirmed by the numerical results. Looking carefully to the analytical and numerical efficiency characteristics, although, one can see that in the case of the new design the numerical efficiency curve is much closer to the analytical one as it is the case for the old design. This means that the new design is performing much closer to its analytical design and hence does not only have a design with higher efficiency but also performs more efficiently in a general sense, i.e. with fewer losses. Generally, one should bear in mind when comparing the analytical predictions with the CFD results that the analytical curves were computed without considering the three-dimensional aspect of the flow, viscosity, turbulence, and flow detachments. Hence, the analytical results represent a theoretical maximum that will never be reached. The good design,

Fig. 17 The blockage effect showing the total unblocked area A_T , the blocked area A_B , and the free area A_F

Fig. 18 Analytical and numerical computation of the relative velocity at different flowrates

although, will have CFD results much closer to its analytical predictions than the poor design. Therefore, all analytical curves are above the CFD results, as it should be. The relevant difference between the old and the new designs is, however, that the CFD results of the new design are much closer to its analytical prediction than the corresponding curves for the old design.

To understand why this is happening, one has to come back to the Carnot principle and have a close look at the relative velocities in the impellers.

In Fig. 18, the relative velocity is shown against radius in the blade channel for different flowrates, computed analytically and numerically. For both designs, the numerically computed relative velocities are higher than the analytical values, as it should be. To understand why this happens, one has to watch how numerical values are computed. They are computed on surfaces of revolution at different radii, in the present case seven surfaces for each impeller (Fig. 19).

The physical quantities are computed as an average of these surfaces at a given radius. There are basically two well-known methods to compute the average of

a generic quantity ϕ : mass flow and area averaging. In the case of the relative velocity w these averaging methods are illustrated in Fig. 17. Two blade channels are marked: the first one, blade channel I, has almost no vortices and the flow is quite unblocked. Disregarding compressibility effects, the relative velocity here corresponds practically to the ratio of flowrate by the total area A_T , i.e.

$$w_I = \frac{Q}{A_T} \quad (56)$$

Assuming first that in both blade channels one has the same flowrate Q , the relative velocity in the second blade channel II at a given radius is given by

$$w_{II} = \frac{Q}{A_F} \quad (57)$$

The blocked area of blade channel II (Fig. 17) is given by

$$A_B = A_T - A_F \quad (58)$$

Substituting equations (56) and (57) into equation (58) and rearranging terms one gets

$$\text{Blockage} \equiv \frac{A_B}{A_T} = 1 - \frac{A_F}{A_T} = 1 - \frac{w_I}{w_{II}} = 1 - \frac{w_{aa}}{w_{mfa}} \quad (59)$$

Fig. 19 Surfaces to evaluate the numerical results as a function of the radius

since, due to the two different averaging methods, the area averaged velocity w_{aa} will always provide the

unblocked velocity w_i and the mass flow averaged velocity w_{mfa} will always provide the blocked velocity w_{II} . In case the blade channel is unblocked, the area averaged velocity w_{aa} is, of course, equal to the mass flow averaged velocity w_{mfa} . As already mentioned in section 7, the mass flow and flowrate can be different from blade channel to blade channel, with differences of up to 5 per cent from blade channel to blade channel in the present case. Applying equation (59) to the whole surface of revolution at a given radius (Fig. 19), one will obtain global information at a given radius about the blockage in the impeller. The blockage is due to flow detachments and vortex formation in the blade channels, as can be seen in Fig. 17. The higher the blockage, the worse the impeller is operating at the given flowrate. Only in the case of ideal flow the blockage is zero. Hence, blockage is also a measure of how far away the impeller is performing from the ideal flow impeller.

The blockage value has been computed for the old and new designs and plotted for several flowrates in Fig. 20. One can see that the new design has much less blockage than the old design, which also explains the better performance of the new impeller as shown in Figs 11 and 12. One can see that especially for flowrates above 50 l/s the blockage of the new design is much lower than that of the old design. That is one reason why the new design is performing substantially better at higher flowrates. At lower flowrates, especially below 30 l/s, the impellers have comparable blockages and

perform quite similarly, as can also be seen in Figs 11 and 12.

Another indicator of the performance of an impeller, as already mentioned in the discussion after equation (43), is the flow angle $\gamma(r)$. The closer the flow angle is to the blade angle $\beta(r)$, the better the impeller is performing. In Fig. 21, the flow angle is plotted against the radius for different flowrates for the old and the new designs. Also, here it is clear that the new impeller is performing much better than the old one since the flow angle is much closer to the blade angle in the case of the new impeller, for all flowrates.

It is important to have a close look at the inlet flow angle, since the bigger the difference between the blade angle and the flow angle at the impeller inlet, the higher the shock losses due to the misalignment of the flow direction and the blade direction. One can see from the left panel of Fig. 22 that the inlet flow angle γ_1 differs much from the inlet blade angle β_1 in the case of the old design, up to $\sim 20^\circ$. In the case of the new design, the deviation of the flow angle from the blade angle is much smaller, up to a maximum of about 5° . Hence the old impeller has considerably higher losses at the impeller inlet than the new design, as also confirmed by the performance characteristics, efficiency, and pressure, in Figs 11 and 12.

Special attention has to be given also to the flow angle at the impeller outlet, since it will determine the power transferred to the fluid according to equations (18) and (21). In the right panel of Fig. 22, these

Fig. 20 Flow blockage of the old and new designs against radius for different flowrates

Fig. 21 Flow angle and blade angle against radius at different flowrates for the old and new designs

Fig. 22 Flow angle and blade angle against flowrate at the inlet and outlet for the old and new designs

values are shown. One can see that in the case of the old design the flow angle at the impeller exit is $\sim 15^\circ$ less than the blade angle for flowrates > 40 l/s, whereas in the case of the new design it is around 5 per cent. This means that there is a reduction in circumferential velocity as given by equation (18) and hence of the total

pressure according to equation (2), which corresponds to the reduction of output as known in the literature, e.g. Eck [2]. It also means that, because the reduction in the outlet angle in the case of the old design is much higher than that in the case of the new design, the reduction of output of the old design is much higher

than that for the new design. This is also one major reason why for the old design the pressure characteristic as computed by the numerical solver is much lower than the analytical curve, and for the new design these two curves, numerical and analytical, are much closer, as can be seen in Fig. 12.

In the previous section, the results of numerical computations were discussed and compared to the analytical predictions. The numerical results already predicted a substantial improvement in performance, i.e. the new design performs better than the old one. To get a final confirmation of the improvements achieved with the new design, a prototype was built and measured on the test rig.

In Fig. 23, both prototypes are shown, the old and the new designs. For a better visualization of the two impellers, a CAD model of the corresponding prototypes with shroud removed is shown in Fig. 24. These impellers were mounted in the very same universal electrical motor drive and inside a spiral casing and measurements were carried out. The test set-up used was a suction side test rig according to the German standard DIN 24 163 part 2. The results of the measurements are presented in Figs 11 and 12. One can see from Fig. 12 that the pressure predictions of the numerical computations agree quite well with the measurements. They do not match perfectly since the impellers were run in a spiral casing, which was not simulated. The spiral casing was not optimized yet and hence the gain in static pressure due to the spiral casing is small and restricted to the mid-range flowrates. Still the improvement in static pressure and efficiency characteristics was predicted well by the

Fig. 23 Prototypes of the old (left) and new (right) designs

Fig. 24 CAD model of the old (left) and new (right) designs with shroud removed

numerical simulation. The total-to-static efficiency was improved by 13 per cent and the maximum flowrate increased by 11 per cent.

Finally, the analytically predicted speed, based on the experimental torque-speed characteristic presented in Fig. 9 and on the theoretical equation (42), was also verified experimentally. The results are presented in Fig. 13. Considering that the analytical speed curves were predicted analytically, the agreement with measurements is very good. In fact, the analytical prediction of speed was the basis for the development of the new design.

9 CONCLUSIONS

By extending the known basic equations of turbomachinery, a new set of mean line equations was derived. These equations, combined with the Carnot principle, are the basis to determine the impellers' main dimensions. Furthermore, it was shown how to also include the torque-speed characteristic of the motor into the design and how to design the blade shapes by an inverse method. Finally, the method was applied to a real case, and it was shown that with one single new design it is possible to achieve substantial performance improvements. In the experience of the authors, who have already designed several different impellers for the industry with this method, this method has proved to be extremely robust, fast, and reliable. With one single new design efficiency improvements of ~13 per cent were achieved. The design method is best used in combination with a commercial numerical flow solver for design verification and analysis. This brings about several advantages: above all, one can confirm and compute the expected performance values such as pressure and efficiency, but further, it allows a detailed flow analysis to track the reasons for performance improvement and to eventually locate potential for further improvements. In combination with numerical computations, this method is best suited to verify the design in an industrial application environment. It can, however, also be used by smaller companies to create very good impellers even without verification by means of a commercial flow solver.

REFERENCES

- 1 Bohl, W. and Elmendorf, W. *Stroemungsmaschinen 1*, 2004 (Vogel Buchverlag, Würzburg).
- 2 Eck, B. *Fans*, 1973 (Pergamon Press, Oxford).
- 3 Eckert, B. and Schnell, E. *Axialkompressoren und Radialkompressoren, Anwendung – Theorie – Berechnung*, 1953 (Springer-Verlag, Berlin).
- 4 Sigloch, H. *Strömungsmaschinen, Grundlagen und Anwendungen*, 2, 1993 (Auflage, Carl Hanser Verlag München).

- 5 **Van den Braembussche, R. A.** Modern turbomachinery component design. In Conference on Modelling Fluid Flow (CMFF'06), Conference Proceedings (Eds T. Lajos and J. Vad), Budapest, Hungary, 6–9 September 2006.
- 6 **Dixon, S. L.** *Fluid mechanics and thermodynamics of turbomachinery*, 4th edition, 1998 (Butterworth & Heinemann, Oxford).
- 7 **Kremser, A.** *Elektrische Maschinen und Antriebe (Grundlagen, Motoren und Anwendungen)*, 2, 2004 (Auflage, Teubner Verlag Stuttgart).
- 8 **Irrgang, R.** Lazare Carnots naturwissenschaftliche Schriften. *Wissenschaftliche Zeitschrift der Technischen Universität 'Otto von Guericke' Magdeburg*, 1989, **33**, Heft 2, 55–60.
- 9 **Douglas, J. F., Gasiorek, J. M., Swaffield, J. A., and Jack, L. B.** *Fluid mechanics*, 5th edition, 2004 (PEARSON Prentice Hall, London).
- 10 **White, F. M.** *Fluid Mechanics*, 4th Edition 1998 (McGraw Hill Higher Education, Singapore).
- 11 **Casey, M. V.** The industrial use of CFD in the design of turbomachinery. AGARD Lecture Series on Turbomachinery Design Using CFD, May to June 1994.
- 12 **Pantell, K.** Über die Schaufelausbildung von Turboarbeitsmaschinen. *Konstruktion*, 1949, Heft 3, S. 77–82.
- 13 **Hönnmann, W.** Zum Problem der optimalen Laufradbreite bei Radialventilatoren, *Heiz.-Lüft.-Haustechnik*, 1961, 12, Nr. 6, 20. Juni, S. 161–162.
- 14 **Sentek, J. and Fortuna, S.** *Modelluntersuchungen an Ventilatoren*. Zeitung Heizung-Lüftung-Haustechnik, 1977, Bd. 28, S. 497 ff.
- 15 **Prandtl, L. and Tietjens, O. G.** *Fundamentals of hydro- and aeromechanics*, 1934 (Dover Publications Inc., New York).
- 16 **White, F. M.** *Viscous fluid flow*, 2nd edition, 1991 (McGraw-Hill Book Co., Singapore).
- 17 **Menter, F. R.** Zonal two equation $k-w$ turbulence models for aerodynamic flows. AIAA paper 93-2906, 1993.
- 18 **Bardina, J. E., Huang, P. G., and Coakley, T. J.** Turbulence modeling, validation, testing and development. NASA Technical Memorandum 110446, 1997 (see also Bardina, J. E., Huang, P. G., and Coakley, T. Turbulence Modeling Validation. AIAA paper 97-2121).

BIBLIOGRAPHY

- DIN-Taschenbuch 254. *Ventilatoren: Normen, Technische Regeln*, 1992, 1. (Auflage, Beuth Verlag, Berlin)
- Epple, P., Ilic, C., and Durst, F.** Combined analytical and numerical radial fan performance optimization. In Conference on Modeling Fluid Flow (CMFF '06), Budapest, 2006, pp. 1110–1117.

APPENDIX

Notation

b	blade height (m)
c	absolute velocity (m/s)
CFD	computational fluid dynamics

d	diameter (m)
M	torque (N m)
n	speed (min^{-1})
n_q	specific speed (min^{-1})
p	pressure (Pa)
P	power (W)
Q	flowrate (m^3/s)
r	reaction (dimensionless)
r	radius (m)
R	radius of circular arc blade (m)
Re	Reynolds number (dimensionless)
s	blade thickness (m)
u	peripheral velocity (m/s)
w	relative velocity (m/s)
x_M	x -coordinate centre of circular arc blade (m)
y_M	y -coordinate centre of circular arc blade (m)
z	number of blades (dimensionless)
α	diffuser blade angle (rad), absolute impeller velocity angle
β	impeller blade angle (rad)
γ	flow angle (rad)
δ	diameter coefficient (dimensionless)
Δ	variation of a quantity (dimensionless)
η	efficiency (dimensionless)
θ	blade angle, wrap angle (rad)
ν	kinematic viscosity of air (m^2/s)
Π	compression rate (dimensionless)
ρ	density (kg/m^3)
σ	speed coefficient (dimensionless)
ϕ	general flow quantity
X	non-dimensional flowrate (dimensionless)
ω	angular velocity (rad/min)

Subscripts and superscripts

1	at the impeller inlet
2	at the impeller outlet
I	blade channel I
II	blade channel II
aa	area averaged
d	dynamic
dif	diffuser
drive	drive
hyd	hydraulic
i	index of cell number
imp	impeller
impeller	impeller
m	meridional
mfa	mass flow averaged
rig	referring to test rig
s	static
shaft	shaft
t	total
u	circumferential